E.T.S. de Ingeniería Industrial, Informática y de Telecomunicación

Design and implementation of an experimental set-up to study vortex induced vibrations

Grado en Ingeniería
en Tecnologías Industriales

Trabajo Fin de Grado

Miguel Hermoso de Mendoza Sarasa

Prof. Dr. Mark Runacres

Ir. Lieven Standaert

Ing. Geert Kermenade

Brussels, 24/06/2016

Abstract

Vortex-induced vibrations (VIV) are periodic motions experienced by a solid due to its interaction with an external fluid flow, which is generally air or water. These vibrations are caused by fluid dynamic oscillating forces which appear during this interaction, causing important fatigue damages in many applications, such as marine cables, bridges or off-shore structures. Thus, VIV is a field of study which is being strongly developed nowadays.

In this project, an experimental set up for the study of water VIV is designed and built. The objective of the plant is achieving a continuous water flow passing a forcedly oscillating cylinder, provoking vortex shedding. To accomplish it, the set-up is designed consisting of four parts: a water closed circuit, a propel system, an external structure and a driving mechanism for the cylinder. The water circuit is composed of two different parts: a rectangular 0.204 m width x 1 meter long x 0.25 meters high duct made with PVC plates and containing a transparent window allowing the view of the vortex shedding; and a pipe system made with 144.6 mm inside diameter PVC pipes, whose function is joining both ends of the rectangular duct and closing the circuit. To move the water, a propel system is made consisting of four 75 mm water propellers, two of them spinning clockwise and the other two spinning counter-clockwise, moved with two brushless motors. The whole circuit is protected and supported by an external structure made with aluminium profiles. Regarding at the driving mechanism, it is a rod mechanism moved by a stepper motor that allows the cylinder moving synusoidally in the cross flow section.

Head losses in the circuit are calculated to select the correct brushless motor. Besides, the forces acting on the cylinder and its influence on the motor torque are studied, and two different automatic controls are done to control the brushless and the stepper motors.

Abstracto

Las vibraciones inducidas por vórtices son oscilaciones periódicas experimentadas por un cuerpo debido a su interacción con una corriente fluida. Estas vibraciones son provocadas por fuerzas fluido dinámicas oscilantes que aparecen durante esta interacción, causando importantes daños de fatiga en muchas aplicaciones, como por ejemplo, cables marinos, puentes o plataformas oceánicas. Así, las vibraciones inducidas por vórtices son un campo de estudio ampliamente desarrollado en la actualidad.

En este proyecto, se diseña y se construye una planta experimental para el estudio de las vibraciones inducidas por vórtices. El objetivo de esta instalación es conseguir que un cilindro sea bañado por una corriente continua de agua, provocando la aparición de vórtices. Para conseguir este objetivo, se diseña una planta compuesta por cuatro partes: un circuito cerrado de agua, un sistema de hélices, una estructura externa y un mecanismo que mueve un cilindro. El circuito de agua consta a su vez de dos partes: un conducto rectangular formado por placas de PVC que contiene una ventana transparente para ver la estela de vórtices, y un sistema de tuberías de 144,6 mm de diámetro interno cuya función es unir los dos extremos del conducto rectangular, cerrando de esta forma el circuito. Para mover el agua, se diseña un sistema de hélices formado por 4 hélices de 75 mm de diámetro, dos girando en el sentido de las agujas del reloj y las otras dos en el contrario. Todo el circuito esta sujetado y protegido por una estructura hecha con perfiles de aluminio. En cuanto al mecanismo, se diseña un mecanismo de barras biela-manivela movido por un motor paso a paso que permite el movimiento sinusoidal del cilindro.

Se calculan las pérdidas de carga en el circuito para dimensionar el motor brushless. Además, se estudian las fuerzas que actúan sobre el cilindro así como su influencia en el par del motor paso a paso, y se implementan dos controles automáticos para el control de ambos motores.

Table of Contents

Drawings:

1. Introduction

1.1. Writer

This Bachelor Thesis has been written by Miguel Hermoso de Mendoza Sarasa, Industrial engineering student of the Universidad Pública de Navarra and Erasmus student at the VUB.

1.2. Professor

This Bachelor Thesis has been assigned and supervised by Mark Runacres, head of the department of engineering technology at the VUB. The daily work has been supervised by Geert Van Kemenad and Lieven Standaert.

1.3. Objective

Vortex-induced vibrations (VIV) are periodic motions experienced by a solid due to its interaction with an external fluid flow, which generally is air or water. Therefore it affects every solid which is in contact with a moving fluid flow, or every moving solid through a fluid. Thus, the field of application is huge and includes many different engineering situations such as offshore structures, aeronautics, bridges, or marine cables application, many of them with a considerable economic impact.

Vortex-induced vibrations may cause important fatigue damage in offshore structures and provoke instability in the flow. Both of these consequences are generally undesirable. Hence, a lot of efforts are being done in order to try to minimize them, and VIV studies have continued to be studied intensively over the past decades.

As do many other universities throughout the world, the Vrije Universiteit Brussel has a department studying this field. This department is the engineering technology department, leaded by Mark Runacres. It also includes many other professors and doctor students, and they, all together, are working on a study about vortex induced vibrations, both produced by a water flow or a wind one. They already have a plant to study VIV caused by a wind flow but they don´t have anything similar for water flows yet. That is why, Mark Runacres has offered me, Miguel Hermoso de Mendoza, a project consisting of the construction of an appliance where they will be able to reproduce and prove experimentally their theoretical study.

This appliance consists of a water closed circuit which has a straight flow tank in which there has to be an oscillating cylinder. This cylinder needs to be moved by a driving mechanism which also has to be designed. Besides that, measuring equipment has to be installed in order to carry out these experiments.

1.4. Requirements

In order to allow the development of accurate experiments, it is necessary to accomplish the indicated task with some requirements:

- The transversal section in the flow tank has to be of 0.2*0.2 m and the flow tank has to measure at least 1 meter long.
- Water is the most convenient fluid to use in this kind of experiments.
- Water flow though the flow tank has to be as laminar as possible.
- Water flow velocity though the flow tank has to be variable and controllable. Beside it has to allow watching the vortex shedding.
- Cylinder oscillation has to be sinusoidal. Furthermore, its amplitude and its period have to be changeable.
- This device is wanted to house force induced vibration experiments.

2. Theoretical Basis

2.1 Vortices. Vorticity

In fluid dynamics, vortices are zones in a fluid where the flow is spinning around an axis of rotation. Some confusion may exist regarding the relation between the mathematical concept of vorticity and the presence of vortices in a fluid. In fact, vorticity can appear with vortices, and some vortices have zero vorticity. I explain this below:

- **Rigid-body vortex**: rigid-body vortices are characterized by their constant angular velocity (Ω=cnt), what means, knowing that $v = \omega * r$, that their particles velocity increases with their distance to the vortex center.
  The vorticity of the flow is, by definition, the curl of the velocity, and it is calculated below:

$$\Omega = (0 \quad 0 \quad \Omega) \qquad\qquad \text{r} = (x \quad y \quad 0)$$

$$u = \Omega\, x\, r = \begin{vmatrix} i & j & k \\ 0 & 0 & \Omega \\ x & y & 0 \end{vmatrix} = i * \begin{vmatrix} 0 & \Omega \\ y & 0 \end{vmatrix} - j * \begin{vmatrix} 0 & \Omega \\ x & 0 \end{vmatrix} + k * \begin{vmatrix} 0 & 0 \\ x & y \end{vmatrix} = (-\Omega y \quad \Omega x \quad 0)$$

$$\omega = \nabla x\, u = \begin{vmatrix} i & j & k \\ \frac{\partial}{\partial x} & \frac{\partial}{\partial y} & \frac{\partial}{\partial z} \\ \Omega y & \Omega x & 0 \end{vmatrix} = i * \begin{vmatrix} \frac{\partial}{\partial y} & \frac{\partial}{\partial z} \\ \Omega x & 0 \end{vmatrix} - j * \begin{vmatrix} \frac{\partial}{\partial x} & \frac{\partial}{\partial z} \\ \Omega y & 0 \end{vmatrix} + k * \begin{vmatrix} \frac{\partial}{\partial x} & \frac{\partial}{\partial y} \\ \Omega y & \Omega x \end{vmatrix} = (0 \quad 0 \quad 2\Omega)$$

  As a result of this calculation, it is possible to observe that, in each point of the fluid flow, the velocity has a tendency to spin around its own axis. The parameter used in fluid mechanics to indicate this phenomenon is **vorticity,** and it is commonly designated by the Greek letter ω.

As I have done before, vorticity can be calculated as the rotational of any fluid particle velocity:

$$\vec{\boldsymbol{\omega}} = \boldsymbol{\nabla} \times \vec{\boldsymbol{u}}$$

$\vec{u}$ = fluid velocity (m/s).

∇ = Nabla operator.

It is important to make the different between angular velocity and vorticity. The first one refers to the rotation movement of the whole fluid around its center of gravity and the second one makes reference to the rotation of each fluid particle around their own axis. It could be understood as the double motion of each solar system´s planet. The orbit around the sun would be angular velocity whereas around the planet´s axis of rotation would be vorticity.

Accordingly, it has been demonstrated that this sort of vortices do have vorticity, and it is the same in every point of the vortex so that the water particles spin around their axes with the same angular velocity.

- **Irrotational vortex or free vortex:** This sort of vortex is characterized by the fact that the product radius per velocity maintains constant in the whole vortex. Thus, velocity decreases with the distance from the vortex center; having the fastest velocity in the rotational axis (it would be infinite).

These vortices are also named irrotational vortices because the vorticity is zero for all their particles:

$$\Omega = (0 \quad 0 \quad Cr^{-2}) \qquad \text{r} = (\,x \quad y \quad 0\,) \qquad r = \sqrt{x^2 + y^2}$$

$$u = \Omega\, x\, r = \begin{vmatrix} i & j & k \\ 0 & 0 & Cr^{-2} \\ x & y & 0 \end{vmatrix} = i * \begin{vmatrix} 0 & Cr^{-2} \\ y & 0 \end{vmatrix} - j * \begin{vmatrix} 0 & Cr^{-2} \\ x & 0 \end{vmatrix} + k * \begin{vmatrix} 0 & 0 \\ x & y \end{vmatrix} =$$

$$= (-Cr^{-2}y \quad Cr^{-2}x \quad 0)$$

$$\omega = \nabla x\, u = \begin{vmatrix} i & j & k \\ \dfrac{\partial}{\partial x} & \dfrac{\partial}{\partial y} & \dfrac{\partial}{\partial z} \\ -Cr^{-2}y & Cr^{-2}x & 0 \end{vmatrix} =$$

$$= i * \begin{vmatrix} \dfrac{\partial}{\partial y} & \dfrac{\partial}{\partial z} \\ Cr^{-2}x & 0 \end{vmatrix} - j * \begin{vmatrix} \dfrac{\partial}{\partial x} & \dfrac{\partial}{\partial z} \\ -Cr^{-2}y & 0 \end{vmatrix} + k * \begin{vmatrix} \dfrac{\partial}{\partial x} & \dfrac{\partial}{\partial y} \\ -Cr^{-2}y & Cr^{-2}x \end{vmatrix}$$

$$= \; 0 * i + 0 * j + k * \left(\frac{\partial}{\partial x}(Cr^{-2}x) - \frac{\partial}{\partial y}(-Cr^{-2}y) \right)$$

$$= k * \left(\frac{\partial}{\partial x} \left(\frac{Cx}{\sqrt{x^2 + y^2}^2} \right) - \frac{\partial}{\partial y} \left(\frac{-Cy}{\sqrt{x^2 + y^2}^2} \right) \right)$$

$$= k * C * \left(\frac{-x^2 + y^2}{(x^2 + y^2)^2} - \frac{-x^2 + y^2}{(x^2 + y^2)^2} \right) = (0 \quad 0 \quad 0)$$

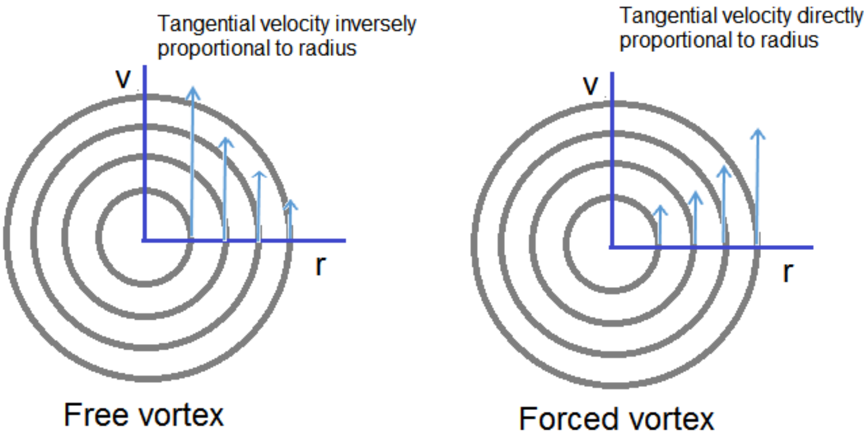


FIGURE 1: FREE AND FORCED VORTEX

Both kind of vortex have their drawback. On the one hand, free vortices are supposed to have an infinite velocity in their centers and, on the other hand, forced vortices need an external force to subsist. That is the main reason why they don´t usually appear separated but together. Thus, the center of vortices usually take the shape of a forced vortex, so that the velocity in the center is 0, while their outskirts behave as free vortices, being also 0 velocity of the furthest part form the center of the vortices [1]. They usually seem like figure 2:

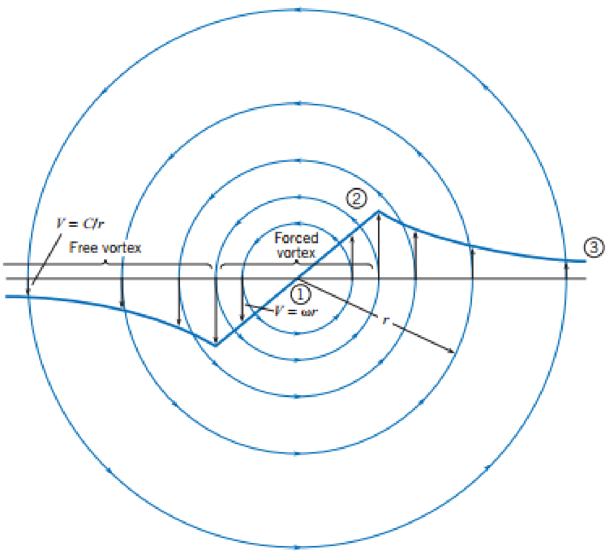


FIGURE 2: COMBINATION OF FORCED AND FREE VORTEX TO MODEL A CYCLONIC STORM.[1]

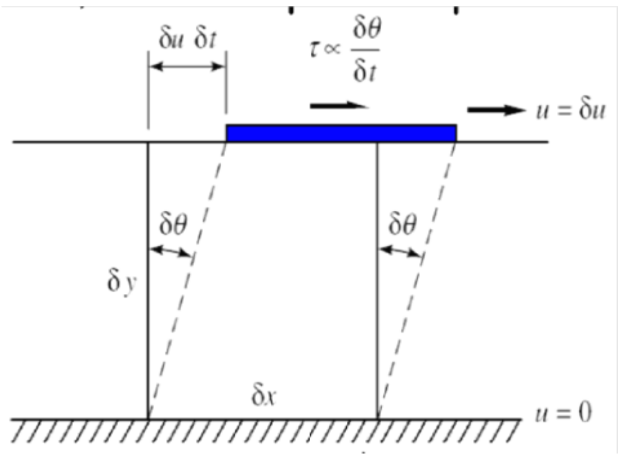
2.2. Vortex-Induced Vibrations

2.2.1. Viscosity, shear stress and boundary layer.

Before starting discussing about the effects of placing a cylinder in a fluid flow, it is worthwhile to introduce some basic fluid dynamic concepts, such as shear stress, viscosity or boundary layer.

In fluid mechanics, when there is a fluid flow streaming through a duct, the shear stress is defined as a force parallel to the fluid movement and to the wall of the duct.

When a fluid is subjected by a shear stress (τ), it starts moving in the direction of the force. Assuming that there is no displacement between the fluid and the walls (no sly condition), the velocity on the cross flow section is not constant but there is a velocity gradient. It is zero on the duct walls and maximum of the center of the duct. Hence, the fluid starts deforming in the surroundings of the walls, as it can be observed in figure 3.



u = Flow velocity (m/s)

$\delta\theta$ = Deformation angle (°)

y = Vertical distance from the wall (m)

x = Horizontal distance (m)

FIGURE 3: FLUID DEFORMATION DUE TO S SHEAR STRESS

In common fluids, such as water or oil, there is a direct relation between the applied stress and the deformation velocity. Thus:

$$\tau \propto \frac{\delta\theta}{\delta t}$$

In addition, this expression can be obtained from the geometry in figure 3:

$$\tan(\partial\theta) = \frac{\delta u * \delta t}{\delta y}$$

In the limit case of infinitesimal variations:

$$\tan(\delta\theta) = \delta\theta$$

$$\theta = \frac{\delta u * \delta t}{\delta y} \rightarrow \frac{\delta\theta}{\delta t} = \frac{\delta u}{\delta y} \rightarrow \tau \propto \frac{\delta\theta}{\delta t} = \frac{\delta u}{\delta y} \rightarrow In\ the\ case\ of\ infinitesimal\ variations \rightarrow$$

$$\tau \propto \frac{d\theta}{dt} = \frac{du}{dy}$$

Therefore, it is easy to observe that the shear stress (τ) is directly proportional to the velocity gradient. The proportionality constant is what is called in fluid dynamics, dynamic viscosity (μ):

$$\tau = \mu * \frac{d\theta}{dt} = \mu * \frac{du}{dy}$$

The fluids that obey this equation for a constant velocity are called Newtonian Fluids, and their velocity field is similar to the one that can be observed in figure 4.

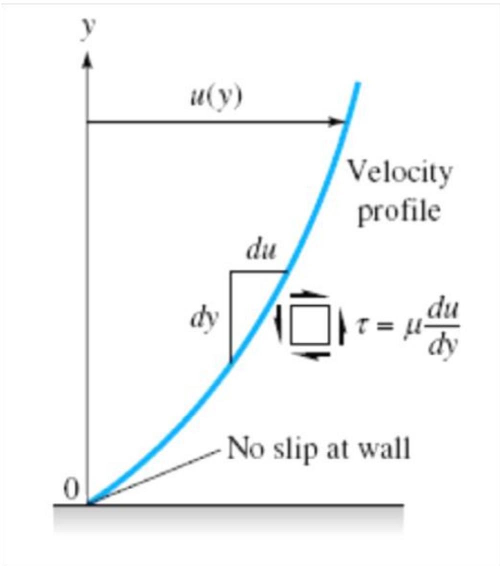


FIGURE 4: NEWTONIAN FLUID VELOCITY PROFILE ON THE SURROUNDINGS OF THE WALL

However, this calculation is only valid in the surroundings of the duct walls. Indeed, as we advance towards the center of the duct, in particular in the presence of turbulence, the fluid viscosity effect decreases and the velocity of the fluid becomes stable. Thus, the fluid flow velocity achieves its maximum value at a certain distance from the duct walls, and it keeps constant until the duct center.

In fluid mechanics, this region of the fluid dominated by viscosity which as a variable velocity is called boundary layer and it can be observed in figure 5.

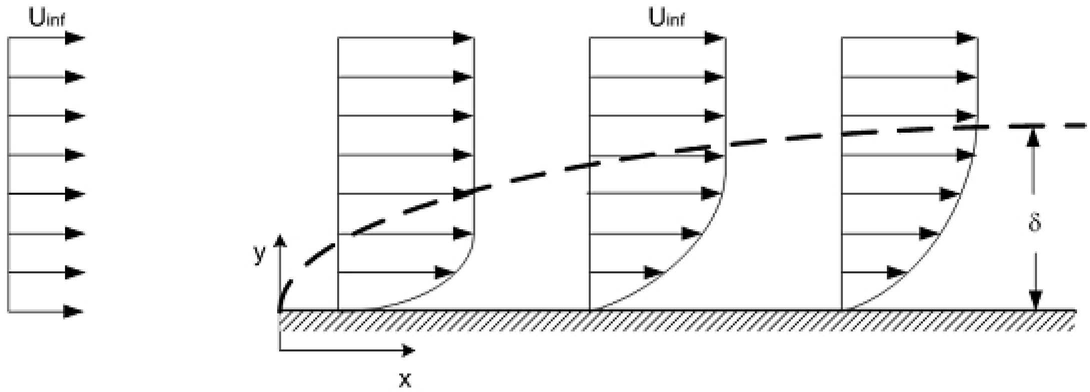


FIGURE 5: BOUNDARY LAYER (THICKNESS = δ) OF A FLUID FLOW IN CONTACT WITH A SOLID WALL.

2.2.2. Flow past a cylinder. Vortices

The behavior of a fluid flow passing a body is relatively difficult to explain. However, assuming that the body is a cylinder, it can be explained by observing the boundary layer.

Ideally, the flow lines are supposed to be like the ones showed in the figure 6. In reality, this pattern only occurs when the Reynolds number is really low (Re≤10). This kind of flow behavior is dominated by viscosity so that the boundary layer does not separate from the body.

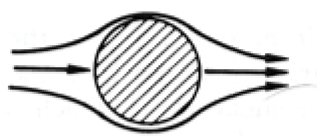


FIGURE 6: IDEAL FLOW LINES PAST A CYLINDER FOR RE≤10 [2]

Otherwise, for higher Reynolds numbers, the flow behavior is markedly different, and it depends on the body´s shape and the fluid energy. The presence of a body (cylinder) on a fluid path influences the flow around it due to its shape. More accurately, it causes a non-favorable pressure gradient behind it. Thus, the cylinder wake depends on the energy of the incoming flow. The adverse pressure difference may slow down the fluid particles in the boundary layer until a separation point, where the fluid flow turns aside. Thereby, the boundary layer separates from the body and there appears a reversed flow which causes a recirculation [3][4], as can be observed in figure 7.

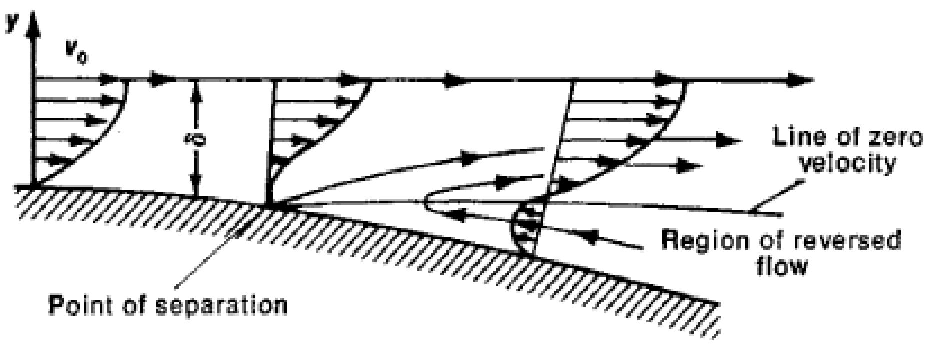


FIGURE 7: SEPARATION OF THE BOUNDARY LAYER FROM A BODY´S SURFACE.

These recirculations are what we call vortices. They can be classified by the fluid Reynolds number, as it can be seen in figure 8. When the Reynolds number is lower than 50, the shed vortices are steady, whereas for Reynolds number bigger than 50, these swirls start becoming instable. Fluid dynamic forces start acting on the cylinder and moving it slightly from one side to the other side on the cross-flow section. These oscillations cause a periodically and stationary vortex shedding [5]. If the Reynolds number continues increasing, there is a non-clear limit where vortices start disappearing, or at least, they are no longer recognizable. This is because high-turbulence fluid flows have enough energy to follow the cylinder shape and prevent separation on its surface, causing a chaotic wake. It also has to be said that the more turbulent is the fluid, the shorter its wake is. [5]

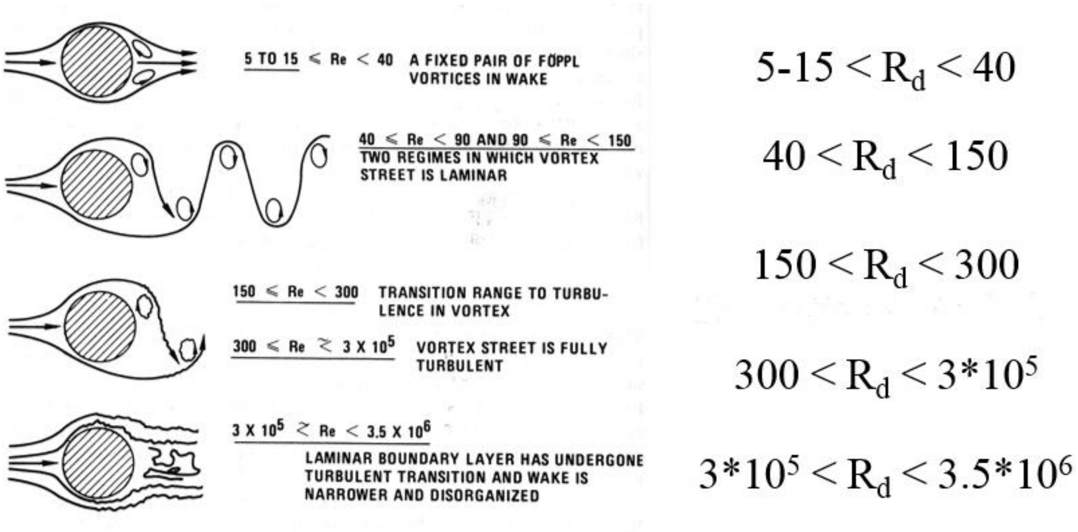


FIGURE 8 : VORTEX SHEDDING DEPENDING ON THE REYNOLDS NUMBER [2]

2.2.3. Strouhal equation. Lock-In

One of the most important characteristics of the vortex shedding is its frequency. It can be estimated by the Strouhal equation, which depends on the flow velocity and the diameter of the cylinder: [2]

$$fs = \frac{S * u}{D}$$

S*=Strouhal number.*

u*=flow velocity. (m/s)*

D*=diameter of the cylinder.(m)*

fs *=vortex shedding frequency. (1/s)*

The Strouhal number is a weak function of the Reynolds number as it is possible to see in figure 9:

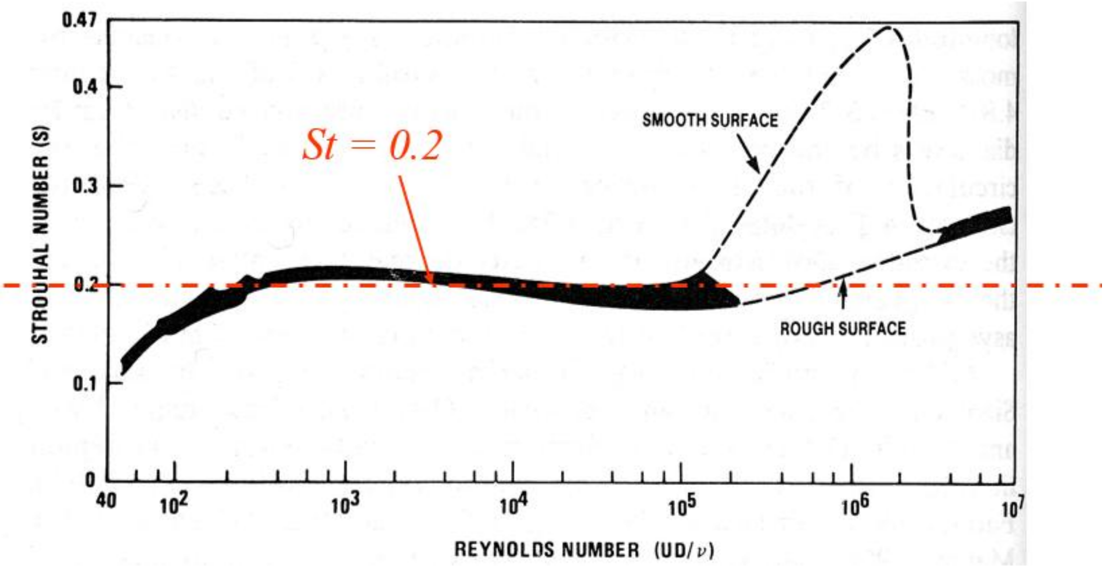


FIGURE 9: STROUHAL NUMBER IN FUNCTION OF THE REYNOLDS NUMBER [2]

Vortex shedding frequency doesn´t need not occur at the natural (resonance) frequency of the submerged structure. However, there exists an interesting natural phenomenon, called Lock-In, which takes place when the ratio between the shedding and natural frequencies is nearby one. When this condition happens, both frequencies fall together near the natural frequency, but not exactly in it (Figure 11). Besides, during lock-in condition, the biggest cross-flow cylinder oscillations can be observed. When there is not this synchronization, the vortex shedding is referred to as non-lock-in. [6][7]

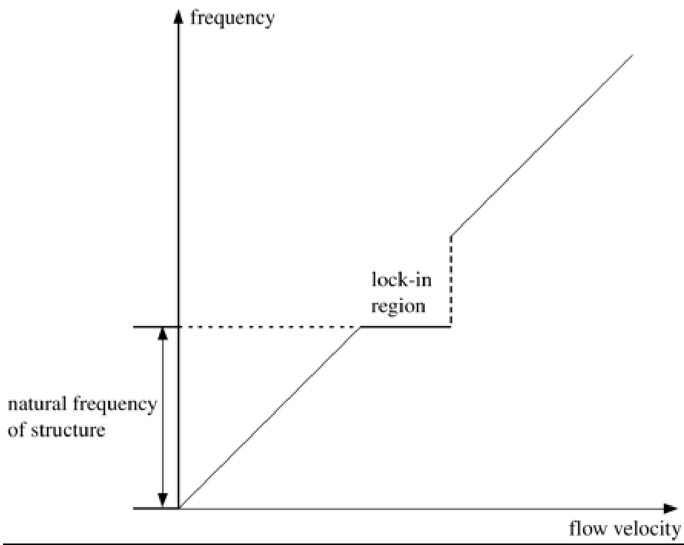


FIGURE 10: VORTEX SHEDDING FREQUENCY. IT CAN BE OBSERVED THE VORTEX SHEDDING FREQUENCY BEHAVIOR CHANGE DURING WHILE LOCK-IN.

2.2.4. Free Vibrations and forced vibrations

During the last decades different experimental investigations of vortex-induced vibrations have been carried out. All of them can be included in one of these two groups:

- **Free induced vibrations:** In this kind of experiment the cylinder is not excited by anything but the water flow. Therefore, it is the water flow which is responsible for all the cylinder movements, helped by the elasticity of the cylinder itself.

  Therefore, this kind of experiment is strongly influenced by the conditions in which they are done. As a matter of fact, a small increase of flow velocity can result in a big change in the vibration amplitude, depending on the proximity of the vortex shedding frequency from the natural frequency. Indeed, during lock in, the amplitude of the vibrations can rise until 0.9 times the diameter of the cylinder. [6]

- **Forced induced vibrations:** Here, on the other hand, cylinder movement is imposed by an external force. There is a driving mechanism that forces the cylinder to move following a fixed function. Therefore, in these experiments, the performance is different to free vibrations, because the forces on the cylinder are not causing the cylinder movement but can be either helping or impeding the movement.

  In this kind of experiment, the shedding frequency and cylinder oscillations frequency don´t have anything to do with each other during non-lock-in conditions. There, the shedding frequency is only related with the water flow velocity though the Strouhal Number, as I have said before. However, during lock-in conditions, the shedding frequency stops depending on the water velocity and falls towards the cylinder oscillations frequency, which acts as its natural frequency.

  The principal upside of the forced induced vibrations experiments is that they are steadier than the free ones, so that it is easier to study them. Besides, forced moving cylinders are able to shed all kind of vortex shedding modes, which will be explained on the next paragraph, while free ones only can shed some of them.

2.2.5. Modes of vortex shedding

During free induced vibrations, it is really usual the shedding of one single vortex each time the cylinder reaches the tops of its oscillation. However, during force induce vibrations, the vortex shedding doesn’t always occur in the same way. Actually, there are four modes of vortex shedding which are commonly related with two parameters:

- Normalized amplitude (A*): It refers to the cylinder oscillations amplitude divided between the cylinder diameter:

$$A^* = \frac{A}{D}$$

- Normalized wavelength (λ*): It refers to the cylinder oscillations wavelength divided between the cylinder diameter:

$$\lambda^* = \frac{\lambda}{D}$$

Thus, depending on the A*: λ* parameter space, we can find the following vortex shedding modes, which can be visualized in figure 11: [8]

- **2S**: It is the most usually mode and it is normally named von Kármán vortex street mode. It consist in the fact that only one vortex is shed in each side of the cylinder´s route.

- **2P**: In this case, a pair of counter-rotating pair of vortices is shed in each side of the cylinder´s route. Both of vortices are equally strong.

- **P+S**: A pair of counter-rotating equally strong vortices is shed in one side of the cylinders path whereas only one vortex is shed in the other side. This mode implies an energy transfer from the cylinder to the water so that it is not expected to find it in the cylinder wake during free induce vibration experiments. [8][9]

- **2Po**: This is the newest mode and it is related with high amplitudes. As it appears in the same region of A*: λ* that the 2P mode, it is commonly so-called the 2P overlap mode. Indeed, it is really similar to the 2P mode but in this case, vortices making up the vortex pair are unequally strong.

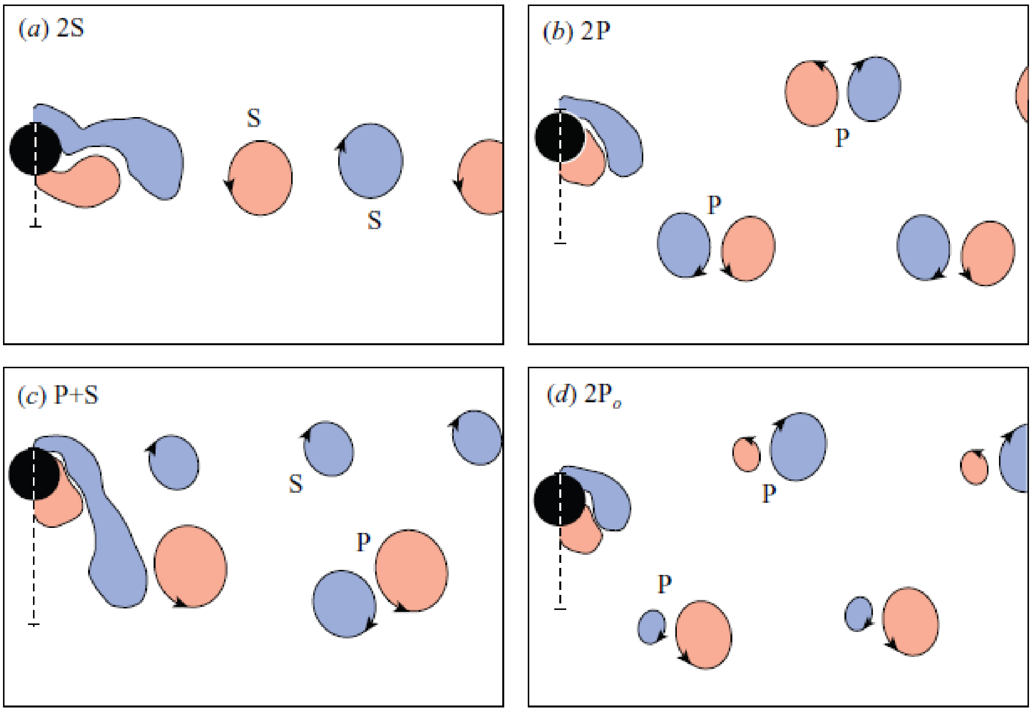


FIGURE 11: VORTICITY FIELDS FOR THE VORTEX SHEDDING MODES: P+S, 2S, 2P, 2PO. [8]

It has been seen that multiple vortex modes could appear in the same region of A*: λ*, so that it is possible that the vortex shedding changes its mode by itself. [8][9]

As I have said before, forced moving cylinders are preferable because they are able to shed any kind of vortex shedding mode, whereas free moving cylinders only can shed the 2S and the 2P modes. Besides, free induced vibrations are to avoid because unpredictable changes on the vortex shedding mode may also change the vibration amplitude without varying the flow velocity. [8]

2.2.6. Forces acting on the cylinder

Understanding vortex shedding phenomenon is quite hard, due to the fact that it is a circular thinking. Mentioning this term, I mean that the cylinder presence causes vortex shedding as well as vortices induce forces on the cylinder, producing the cylinder movement. These forces are mostly due to viscous shearing stresses and pressure gradients, and they can be gathered in two groups, depending on their direction:

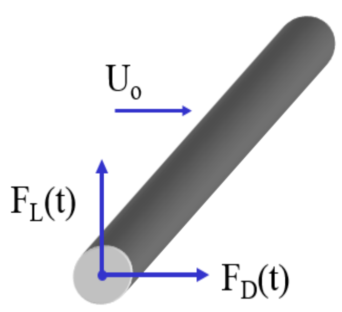
- **Lift force:** This force appears because of the interaction between the water flow and the cylinder and it is the responsible for the vibration of the cylinder and the stationary vortex shedding. Thus, its direction is perpendicular to the stream and it is a pulsating force (in the case of a cylinder). That is due to the fact that it depends on the lift coefficient (C_L), which behaves as a sinusoidal function, as can be observed in figure 14. The lift force is given by the equation below:

$$F_L = \frac{1}{2} * C_L * \rho * u^2 * S$$

C_L= Lift Coefficient
ρ= Fluid density ($\mathrm{kg/m^3}$)
u= Fluid Velocity (m/s)
S= Projected frontal surface (In this case: cylinder length per cylinder diameter) ($\mathrm{m^2}$).

The behavior of the lift force and its influence on the cylinder depend on whether the vibrations are free or forced and whether they are made below lock-in conditions or not.

On the one hand, below the conditions of free vibrations or forced vibrations during lock-in, and the usual vortex shedding mode (2S), the cylinder behaves in this way in relation with the vortex shedding and the C_L: I start by supposing that the cylinder is placed in the lowest position of its motion. It sheds a vortex which pushes the cylinder upwards (C_L positive). When it reaches the top position, it sheds another vortex, with opposite vorticity and a new one starts being formed. This new vortex in formation pushes the cylinder down and the becomes negative. It descends until the lowest position, when the vortex is shed and the cycle commences once more. As a consequence, in these cases the lift force frequency concurs with the cylinder oscillations frequency.



C_L

FIGURE 12: LIFT FORCE AND DRAG

On the other hand, during forced vibrations, if the vortex shedding mode is different or the lock-in condition is not met, the lift force fluctuation has nothing to do with the cylinder frequency. In this case, this force, acting on the cross-flow plane, varies between two senses with a different frequency of the cylinder movement. Hence, it is sometimes sustaining the cylinder oscillations or obstructing them. [8]

- **Drag force**: Drag force is produced by the incoming flow and its direction is parallel to the flow. It includes every force which pushes the cylinder downstream and it is given by the equation below:

$$F_D = \frac{1}{2} * C_D * \rho * u^2 * S$$

C_D= Drag Coefficient.
ρ= Fluid density (kg/m^3).
u= Fluid velocity (m/s).
S= Projected frontal surface (In this case: cylinder length per cylinder diameter) (m^2).

It can be observed that the drag force depends on the density, the squared velocity, the projected frontal area of the body (cylinder) and a mathematic coefficient, which is called drag coefficient (C_D). This coefficient is the factor that fixes the behavior of the drag force. Thus, the C_D, and accordingly, the drag force, both of them behave as a sinusoidal wave whose average is different to zero, as can be seen in figure 14. Thus, the drag coefficient can be decomposed in two terms: [10]

  - Parasite Drag (C_{Do}): This is the constant component of the drag force, causing the fact that it has an average value distinct to zero in the direction of the flow. It refers to almost all the forces acting against the relative motion of the body (cylinder) respecting to the fluid. It is related with the body shape and the Reynolds number, as it can be observed in figure 13.

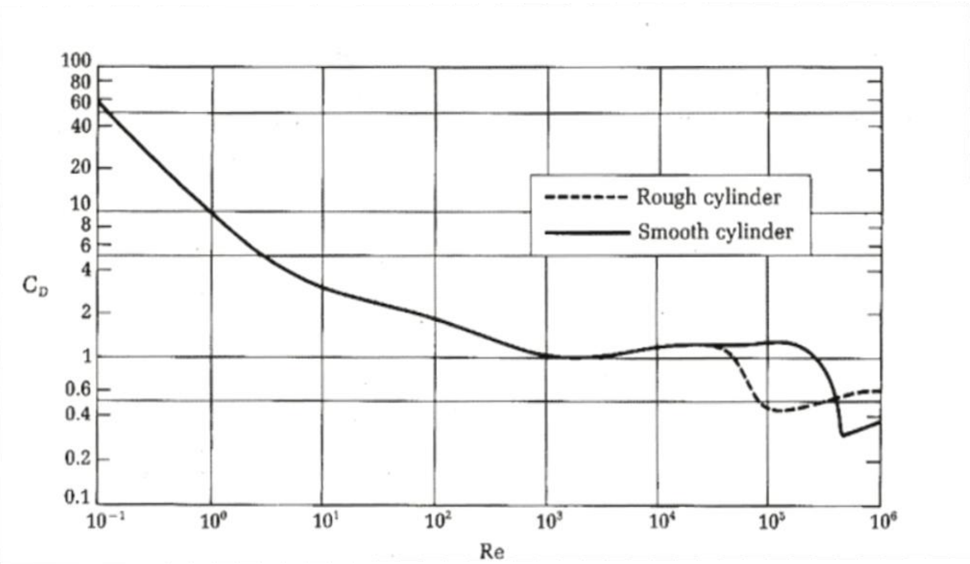


FIGURE 13: DEPENDENCE OF THE DRAG COEFFICIENT ON THE REYNOLDS NUMBER.

As can be observed in figure 13, when the Reynolds number increases, the drag coefficient decreases. It can be explained regarding at the boundary layer and the cylinder wake. The more turbulent the water flow is, the more energy it transports, and the more energy can be put into the boundary layer. Thus, turbulent flows have more energy in their boundary layer than laminar flows, so that their boundary layer have a bigger ability to adapt itself to the shape of the incoming body (cylinder), and therefore, its separation from the body walls is delayed, producing smaller wakes, and accordingly, less drag.

- Induced Drag (C_{Di}): This component of the drag coefficient is to blame for its sinusoidal behavior. As its name indicates, it is caused by the vortex shedding and it depends on lift coefficient. This coefficient varies from 0 to a maximum value, always acting parallel to the direction of the fluid. Its peaks take place when the lift force takes its biggest values, no matter which sense, whereas C_{Di} values zero when the lift force is zero. Consequently, a period of the C_{Di} corresponds to half a period of the lift force.

  It can be calculated with the expression below:

$$C_{D_I} = \frac{{C_L}^2}{\pi * e * AR}$$

e= Oswald efficiency number ($0.6 < e < 0.99$)

AR=Aspect Ratio (in the case of a cylinder $AR = L/D$)

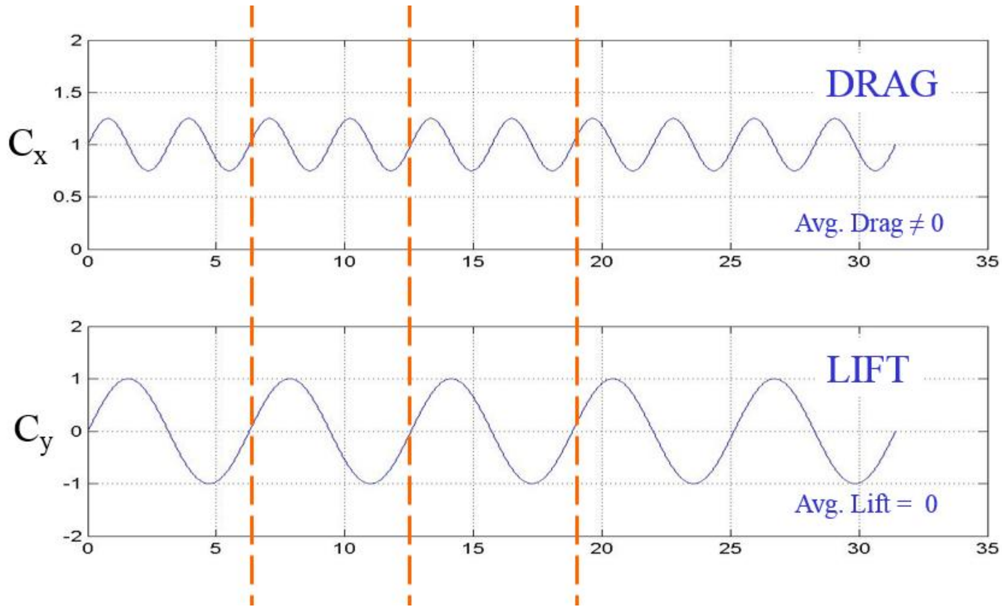


FIGURE 14: SINUSOIDAL VARIATION OF LIFT AND DRAG COEFFICIENT [2]

3. Water circuit

In order to carry out this project and to make possible the conducting of more experiments on this matter, I have to design a water circuit. There are some requirements that have to be done in order to be able to make accurate experiments. Firstly, the circuit has to have at least a flow tank which contains an oscillating cylinder. In addition, the measurements of the tank have been specified: the cross section of the flow rate has to be 0.2 meters wide per 0.2 meters high. These dimensions are the minimum distances necessary to keep constant the velocity in the center of the tank and maintain if far from the effect of the walls. In addition, the tank has to be at least 1 meter long so that the flow has enough distance to become as laminar as possible before reaching the cylinder and that the vortex shedding could be observed after it. Secondly, it has to be possible to carry out a continuous experiment. Therefore, the design has to be a closed circuit. And finally, the design has to be done in a way that permits the visualization of the vortex shedding. As we will see later, this will influence the selection of the materials of the tank, and it will restrict the velocity of the water.

Basically, taking into account all the requirements that were given, I have reached a particular design whose global dimensions are: 1.5 m long, 0.7 m wide and 0.35 m thick. This design can be seen in figure 15, and it is composed of 4 parts; each of them is going to be thoroughly described below:

- **Flow tank:** This is the main part of the project. It is a rectangular box opened on its top so that it form a large duct which is divided in an opaque zone and a transparent window, where it is going to be possible to observe the vortices.

- **Propellers:** Pumps and/or water propellers are essential components in a system based on water circulation because they give to the water the required energy to overcome head losses. In this design, I am going to use boat propellers, as it is going to be explained afterwards.

- **Pipes:** Pipes are only used to connect the beginning of the flow tank to its end, so that the circuit is closed. Their dimensions must be adjusted so that they fit the design guaranteeing a good balance between the head losses they cause and the fact that the water flow must be as laminar as possible.

- **Frame:** It is a structure of aluminum profiles which surrounds the flow tank and the pipes and it has two functions: it bears the forces due to the pressure of the water in the flow tank, preventing the acrylic plates of bending; and it is thought to be the support where both the pipes and the flow tank are placed.

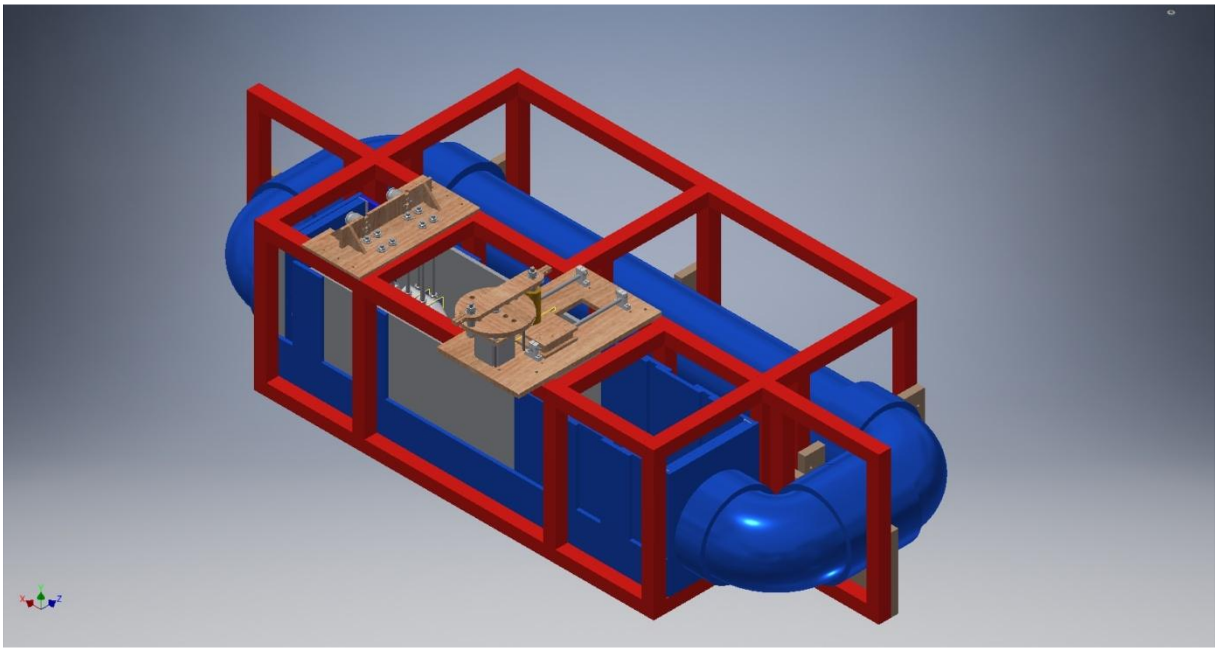


FIGURE 15: WATER CIRCUIT DESIGN. PAINTED IN DIFFERENT COLORS, IT CAN BE SEEN THE WATER CIRCUIT (BLUE), THE FRAME (RED), AND THE DRIVING MECHANISM AND THE PROPELLERS STRUCTURE (BROWN)

3.1. Flow Tank

As I have said before, geometrically, the flow tank is a rectangular box open at the top, as it can be observed in figure 15. It is the main part of the circuit because it is there where vortices are going to be shedding, but it is also the most difficult one, due to the fact that I have to accomplish a laminar flow passing over a cylinder. Flowing water tends naturally to become turbulent at high velocities. Each obstacle the flow came across also contributes rising the turbulences in the fluid. Thus, my design has to allow the flow to be as smooth as possible. Therefore, has to contain the least amount of obstacles possible and when they are absolutely necessary, the less sharp are selected. Nevertheless, despite this fact, water flow may be turbulent. That is the reason why I decide to put two holes in each wall of the box so that meshes can be placed there, standing in front of the incoming flow. These meshes would contribute ordering the fluid flow by separating it in many smaller flows. Initially, I don´t place meshes, as I don´t know how the flow behavior is going to be, but I do give the option to put them to experimenters, in case they need them.

The flow tank is going to be full with water. Therefore, its walls are going to be subjected to forces due to hydrostatic pressure. This hydrostatic pressure is bigger as we go down the wall so that its highest part of the wall is loaded with atmospheric pressure whereas its lowest part is subjected to the atmospheric pressure and the pressure made by the water column. Hence, there is going to be a differential force pushing against the walls of the flow tank. These forces can be reduced to a force whose value is equal to the hydrostatic pressure in the gravitational center multiplied by the area of the plate and which is going to be applied in the center of pressure which is usually located under the gravitational center.

3.1.1. Material

The loads on the flow tank's walls place severe constraints on the wall thickness. There is a minimum thickness necessary to bear this pressure and it depends on the dimensions of the fluid tank and the material from which it is made. Firstly, I have to think about the flow tank's material before thinking in the thickness of its walls. Thus, the first material which comes to my head, steel, has to be discarded because of the simple reason that it is going to rust under a continuous water flow. In spite of this, the flow tank can be still made with stainless steel, plastic or aluminum, which produced an oxidized layer which protects itself against the rustiness. Nevertheless, although there are metallic materials which resists against corrosion in different atmospheres, all of them get hastily rusted when there is another metal in the design. This is due to galvanic corrosion. According with this theory, when there are two metals connected by an electrolyte (in this case the water), a redox-reaction takes place, and consequently, the metal with the lowest reduction potential gets quickly rusted (anode) while the other one gets reduced (cathode). Hence, as the propellers are made in a metallic material, I reject every other metallic material in the construction of the flow tank. Accordingly, there only remain plastics as possible material to carry out the project with.

There were plenty of aspects to take into account in this election, such us resistance, roughness or prices. Nevertheless, the selected material has to have one special characteristic which makes it more interesting than its competitors, which is the fact that it has to be transparent so that it allows us to watch the vortex shedding. That is why, finally, acrylic plastic or transparent PVC would be flow tank material good options.

However, there transparent plastics are more expensive than other opaque plastic. Furthermore, it is not necessary at all that the entire flow tank is transparent. Indeed, it is only absolutely required where the vortex shedding occurs. Therefore, I decide to divide the flow tank two parts: one made on a transparent material and another which can be made with any other opaque material.

- **Opaque part:** Thinking about the other part of the flow tank, I came to the conclusion that is very complicated to made a single piece which could contain the meshes and could be connected with the pipes and the acrylic plates. That is why I decided it to be composed of several plates, as the easiest solution regarding at its construction.
  Regarding at the material, I need a plastic that offers us good mechanical properties, and a simple way of joining it. Thus, PVC is selected because it has notable properties, as it can be looked up on the table 1, and it result a strong easy bond from gluing two PVC pieces.

Property	Value
Density	1380 kg/m³
Young's modulus	2900-3300 MPa
Tensile strength	50-80 MPa
Elongation @ break	20-40%
Impact strenght	2-5 kJ/m²
Glass temperature	87 °C
Melting point	212 °C
Vicat temperature	85 °C
Heat transfer coefficient	0.16 W/m.K
Linear expansion coefficient	8.10^{-5} /K
Specific heat	0.9 kJ/(kg·K)
Water absorption	0.04-0.4

TABLE 1: PVC PROPERTIES [11]

- **Transparent part**: this is the part of the tank in which is the cylinder and in which vortices are going to be shedding so that people, who are experimenting, could watch their experiment's results. It only consists of two transparent plates glued to the other parts of the flow tank. Regarding at the material of the plates, finally, transparent PVC is selected over acrylic due to the fact that PVC-PVC glued joints are strong whereas the glued joints between PVC and other plastics (acrylic) is not that strong.

3.1.2. Design

Once I have decided that the design is composed of several plates, I have to set how they are going to be placed. As I have said before, all of the joints are made gluing the plates. Nevertheless, every way of gluing isn't useful, as the design has to be completely watertight. In fact, as it can be watched in figure 16, there are three main ways of gluing. The third one (lap joint) gives stronger and steadier joins than the other two (butt joint and T joint), basically because it can cover much more space than the other ones, whose maximum thickness is equal to the width of the plates [12][13]. Thus, the lap is best suited to prevent leakage. That is the main reason why, in this design, I always try to use only lap joins when it is possible, and if it is impossible, I protect butt joins with other PVC plates joined rightly.

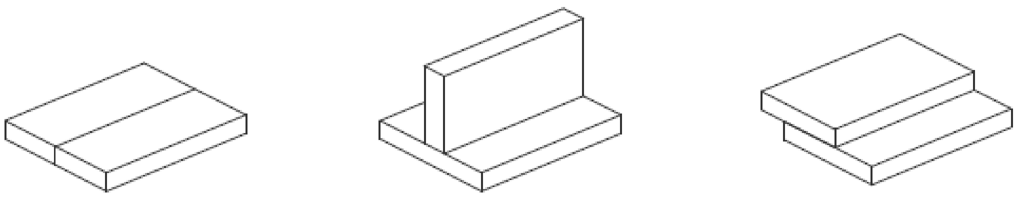


FIGURE 16: BUTT, T, AND LAP JOINTS (FROM THE LEFT TO THE RIGHT)

Regarding at the design, it is composed of a large PVC plate, which is going to act as the floor of the flow tank, and multiple plates that are going to act as walls, as it can be observe in the drawings. In addition, there are external plates placed in every union line whose function is preventing water leakage through these “weak” zones. Regarding at the interior, I put two holes in each wall aimed to harbor meshes if necessary, as I said before. In order to protect them to leakage, I embed in each hole a small 1 mm plate.

Finally, two circular pipes are going to enter the rectangular tank through either of its ends. As the cross section of the tank is squared and the cross section of the pipes is circular, there have to have at least a plate acting as a cap, covering the extra surface of the flow tank cross section, so that the water doesn’t go out. Ideally, these pieces should have a shape that allows a gradual transition between the circular cross section and the squared one. However, these theoretical pieces are quite complicated to build. That is the main reason why, finally, these pieces are going to be plates too, with a hole in the center so that the pipes can come through them. This decision leads some turbulence and head losses due to the abrupt change of the section of the duct. Thus, in order to minimize these side effects, I choose the biggest pipes that can be placed inside the cross section of the tank, decreasing the differences of the ducts sections. In addition, after the entrance of the water to the duct, it faces two meshes, which reduces considerably the turbulence of the fluid, as it is going to be explained before.

After putting a cap in each side of the flow tank, another issue appears. It is possible to perceive on the drawings that the gluing between the caps and the pipes entering through them is going to be a butt join (the undesirable one) due to the tank geometry. Hence, to make up for it, I place in each side an extra plate, so that there is no longer one cap in each side, but two of them. This measure leads to the fact that there are more joined surface between the pipes and the caps.

In addition, the floor of the flow tank has also a drain to allow the tank to empty. This drain is situated after the cylinder, because, as it is a little obstacle that makes turbulences in the flow, it is preferred to be placed after the vortex shedding where it is less important how the flow is.

3.1.3. Thickness

The next step is the calculation of the thickness of the walls. To do it, firstly I have to identify the forces acting on the tank. The only forces acting are hydrostatic pressure and dynamic pressure due to the velocity of the water (Bernoulli’s Equation). As the flow tank width (0.2m) is much smaller than its length (1m), I can neglect the effect of the pressure on the direction of the water flow and focus on the pressure on the lateral walls. Hence, I can also neglect the effect of the dynamic pressure, owing to the fact that it is only perceived on the water flow direction. As consequence of this assumption, I can simplify the forces acting on the tank to the hydrostatic pressure acting on its lateral walls.

Hydrostatic pressure is the pressure caused by accumulated still water. It pushes outwards to every wall of its container and it is directly proportional to the depth, as it can be seen in figure 17. Thus, it only depends on the density of the fluid and depth of the point where the pressure is calculated.

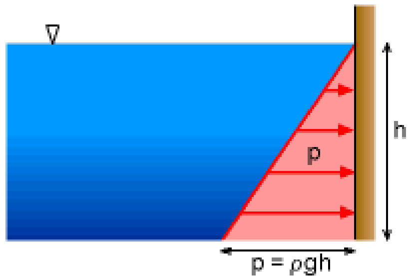


FIGURE 17: HYDROSTATIC PRESSURE.

For the realization of this calculation, I assume that the flow tank laterals consist only of a single 0.2 height x 1 m width plates. In order to calculate the effect of hydrostatic pressure on an immersed plate, it can be considered as a single force applied on the centre of pressure. The modulus of this force is equal to the hydrostatic pressure on the gravity centre of the plate per its area:

$$F_{GC} = P_{GC} * A = \rho * g * h_{CG} * A$$

F_{GC} = Force equivalent to hydrostatic pressure (N).

P_{GC} = Hydrostatic pressure on the gravity center (Pa).

A = Plate area (m^2)

In this case, the gravity centre is on the centre of the plate due to symmetry. Thus, knowing that $h_{CG} = 0{,}1$ m, I calculate the force acting on either of the tank lateral plates:

$$F_{GC} = \rho * g * h_{CG} * A = 988 * 9.8 * 0.1 * 0.2 * 1 = 195.6 \text{ N}$$

To finish this simplified model of the hydrostatic pressure, I have to calculate the center of pressure. Regarding at its horizontal position, it is situated on the centre of the plate due to the plate symmetry. Vertically, it is beneath the centre of gravity. To calculate the vertical distance between both of them, I use the following equation:

$$y_{CP} = -(\rho * g) * \sin\varphi * \frac{I_{xx}}{P_{CG} * A} =$$

$$= -\sin\varphi * \frac{I_{xx}}{h_{CG} * A}$$

I_{xx}=2nd grade Inertia´s moment (horizontal axis)(m^4)

φ= Angle between the plate and the horizontal plane.

A= Plate area (m^2)

P_{CG}, h_{CG}= Pressure and height on the gravity centre (Pa, m)

Thus, I can calculate the distance between the gravity centre and the pressure centre knowing that the angle φ is 90˚, and the inertia of the plate for the axis x is:

$$I_{xx} = \frac{1}{12} * b * h^3 = \frac{1}{12} * 1 * 0.2^3 = 0.00066\, m^4$$

$$y_{CP} == -\sin\varphi * \frac{I_{xx}}{h_{CG} * A} = -\ \sin 90 * \frac{0.00066}{0.1 * (0.2 * 1)} = -0.033$$

Hence, the centre of pressure is situated at a distance of 0.133 m beneath the free surface of the water, and the hydrostatic pressure can be trivialized as a force of 195.6 N acting on this point and pointing outwards.

Regarding at the geometry and the construction of the flow tank, there are two weak points: the glued joints on the extremes of the lateral plates or their center. On the one hand, I calculate the minimum thickness of the glued joints to resist the hydrostatic pressure. In this case, it is acting on the glue joint as a shear force. The shear strength of PVC glues is usually around 4-5 MPa. Therefore, I equal the hydrostatic force to the shear strength of the glue joint per its area. Knowing that the length of the glued union is 0.2 m (the height of the tank):

$$F = A * \tau = t * l * \tau$$ t= glued joint thickness (m)

$$195.6 = t * 0.2 * 4000000 \rightarrow t = 0.245\, mm$$ l= glued joint length (m)

As it can be observed, this distance is very small. Indeed, whether it can support the forces or not, the width of the glued union has to be much bigger to ensure that plates´ joints are watertight.

On the other hand, the centers of the lateral plates are another weak point. This is due to the fact that the water pressure is going to push the plates outwards, and, as they are glued on their ends, they are going to bend outwards. Therefore, the whole plate is going to be affected by a bending moment. Thus, the minimum plate thickness is going to be calculated so that the bending distance between the original point and the deformed one doesn´t surpass a certain value.

In order to analyze the effects of this moment and to size the correct plate thickness to withstand it, I simplify the geometry to a beam supported in both of its ends, neglecting the glued union at the bottom of the plate. To understand this simplification it is enough with looking at the lateral plate from a top view. It can be imagined something like similar to the figure 18, where P=F_{GC}, and, A and B are the glued unions:

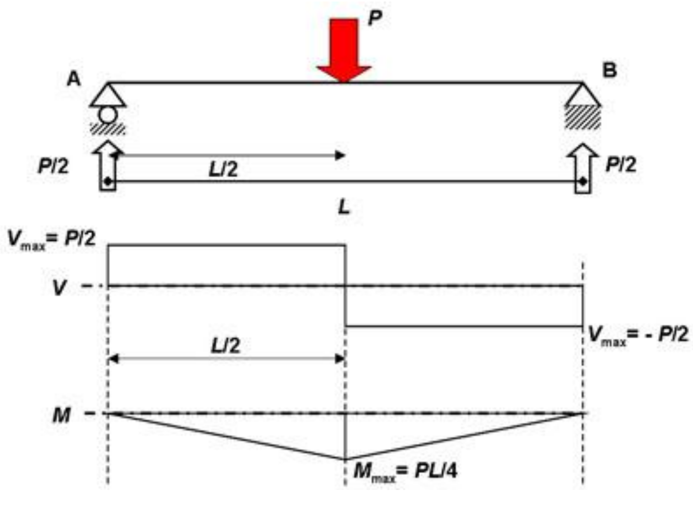


FIGURE 18: SIMPLIFICATION OF THE PLATE BENDING

Due to symmetry, the reactions to the pressure force are going to be equal in both ends of the plate and they are going to be $F_{GC}/2$ N. To simplify the notation, in the following calculation F_{GC} is going to be substituted by F.

The bending moment on the beam can be expressed as a function of the distance for the point A to the point where the bending moment is calculated:

$$\begin{cases} If\ x \leq \dfrac{L}{2} \rightarrow M(x) = \dfrac{F}{2}x \\ If \dfrac{L}{2} \leq x \leq L \rightarrow\ M(x) = \dfrac{F}{2}x - F\left(x - \dfrac{L}{2}\right) = \dfrac{F}{2}x - Fx + \dfrac{FL}{2} \end{cases}$$

The deformation on any point of a beam is given by this equation:

$$EI\frac{d^2y}{d^2x} = M(x)$$

y= deformation on the vertical axis

E= Young Module (Pa)

I = Moment of Inertia (m^4)

As the bending moment function is not expressed by a single law, but two, I have to integrate his two different fields:

$$\boldsymbol{If\ x \leq \frac{L}{2}} \rightarrow EI\frac{d^2y_L}{d^2x} = \frac{F}{2}x\ \rightarrow EI\frac{dy_L}{dx} = \frac{F}{2}*\frac{x^2}{2} + C_1 \rightarrow EIy_L = \frac{F}{4}*\frac{x^3}{3} + C_1x + C_3$$

$$EIy_L = \frac{Fx^3}{12} + C_1x + C_3$$

$$\boldsymbol{If\ \frac{L}{2} \leq x \leq L} \rightarrow EI\frac{d^2y_R}{d^2x} = \frac{F}{2}x - Fx + \frac{FL}{2}\ \rightarrow EI\frac{dy_R}{dx} = \frac{F}{2}*\frac{x^2}{2} - F*\frac{x^2}{2} + \frac{FL}{2}*x + C_2 \rightarrow$$

$$EIy_R = \frac{F}{4}*\frac{x^3}{3} - \frac{F}{2}*\frac{x^3}{3} + \frac{FL}{2}*\frac{x^2}{2} + C_2x + C_4 \rightarrow EIy_R = -\frac{Fx^3}{12} + \frac{FLx^2}{4} + C_2x + C_4$$

Now, I apply the following boundary conditions to obtain the values of the constants that have appeared:

$$\begin{cases} In\ x = L/2 \rightarrow \dfrac{dy_R}{dx} = \dfrac{dy_L}{dx} \\ In\ x = \dfrac{L}{2} \rightarrow y_R = y_L \\ In\ x = 0 \rightarrow y_L = 0 \\ In\ x = L \rightarrow y_R = 0 \end{cases} \rightarrow \begin{cases} C_1 = \dfrac{FL^2}{8} + C_2 \\ C_4 = \dfrac{FL^3}{48} \\ C_3 = 0 \\ C_4 = -\dfrac{FL^3}{6} - C_2L \end{cases} \rightarrow \begin{cases} C_1 = \dfrac{FL^2}{8} + C_2 \\ C_4 = \dfrac{FL^3}{48} \\ C_3 = 0 \\ C_2 = -\dfrac{9FL^2}{48} \end{cases}$$

Resulting from the distribution of bending moment, the maximum deformation of the beam is on its center. Hence, once I have the values of the constants and knowing, I calculate the deformation on the center of the beam as:

$$EIy_R = -\frac{Fx^3}{12} + \frac{FLx^2}{4} + C_2x + C_4 \rightarrow EIy_R = -\frac{Fx^3}{12} + \frac{FLx^2}{4} + -\frac{9FL^2}{48}x + \frac{FL^3}{48}$$

$$In\ x = \frac{L}{2} \rightarrow EIy_R = -\frac{F(\frac{L}{2})^3}{12} + \frac{FL(\frac{L}{2})^2}{4} + -\frac{9FL^2}{48}\frac{L}{2} + \frac{FL^3}{48} = -\frac{FL^3}{96} + \frac{FL^3}{16} - \frac{9FL^3}{96} + \frac{2FL^3}{96} = -\frac{FL^3}{48}$$

$$y = -\frac{FL^3}{48EI}$$

Knowing that the inertia of the plates in this direction is $I = \frac{1}{12} * h * t^3$, and the young modulus of the PVC is 3.1 GPa, I can calculate the thickness of the plate so that the bending distance remains below 0.5 mm. Thus:

$$y = \frac{F * L^3}{48 * E * I} \rightarrow 0.5 = \frac{195.6 * 1^3}{48 * \left(\frac{1}{12} * 0.2 * t^3\right) * 3.1E9} \rightarrow t = 5.40mm$$

Applying a security factor of 1.5: $t = 1.5 * 5.40 = 8mm$

Therefore, I will need 8mm plates to bear the bending moment without surpassing the 0.5 mm limit bending distance that I have imposed. Actually, as it can be read on the beginning of the document and on the paragraph taking about the frame, I put a frame in my design in order prevent this bending moment, due to the fact that it causes the plate bending, as it have been seen right know, and above all, to protect the glued unions from the bending moment, because they are really weak to this kind of load, as it concentrate the tension one edge of the glued union. Nevertheless, in spite of the fact that there the frame is reducing the impact of the bending moment, there is still a long distance on the center of the tank between the two frame columns, where the effects of the bending moment aren´t going to be negligible. Therefore, in order to be sure of the resistance of the plate, I am going to maintain the calculated thickness, even if it could be smaller. In addition, the bigger the glued union thicknesses are, the more watertight they are. Thus, form this sight it is also positive maintaining the 8 mm calculated plate thickness, as it ensures too the absence of water leaks.

3.1.4. Flow velocity

Another important parameter in this design is the flow velocity. It is very important because it is directly proportional to the vortex shedding frequency during non-lock-in conditions. They are related by Strouhal equation, as it has been said before:

$$fs = \frac{S * U}{D}$$

The Strouhal number (U) depends on the Reynolds number, as it can be seen in figure 9. Thus, in order to calculate it, I have to estimate the Reynolds number of the fluid flow passing over the cylinder. In this case, the characteristics length of the Reynolds equation is the distance from the entrance of the tank to the cylinder (x=0.4 m). Thus:

$$Re_x = \frac{u * x * \rho}{\mu}$$

Re_x = Reynolds number for characteristic length L=x

x=Distance from the starting of the plate to the point where Re is calculated (m)

u = velocity of the water (m/s)

$$Re_x = \frac{u * x * \rho}{\mu} = \frac{0.45 * 0.4 * 998}{0.0011} = 163309$$

Knowing that the Reynolds number of the flow passing over the cylinder is going to be 163309, I take the value of the Strouhal number from the graphic of the picture 9. It is accurate enough to take the Strouhal number as a constant: S=0.2, taking into account that the scale of the graphic is logarithmic. On the other hand, the diameter of the cylinder is thought to be variable, as I will explain later. Initially, in order to do the calculations, I should select a cylinder with a representative diameter for every case. I choose a 3 cm diameter as the average value, because a much bigger cylinder wouldn´t be useful due to the width of the tank; and it couldn´t be much smaller too, because vortices wouldn´t appear or wouldn´t be recognizable. Hence, only two free parameters remain: velocity and shedding frequency, whose relation for the desired shedding frequencies is shown in table 2:

Frequency (Hz)	Velocity (m/s)
1	0,15
1,5	0,225
2	0,3
2,5	0,375
3	0,45

TABLE 2: WATER FLOW VELOCITY FOR EACH VORTEX SHEDDING FREQUENCY.

Therefore, the flow velocity is not wanted to be more than 0.45 m/s and the propellers have to be calculated in order to achieve this velocity.

3.2. Pipes

3.2.1. Head losses introduction. First grade and second grade losses

All kinds of fluid motions cause head losses. These losses depend on the surface roughness, the duct geometry and above all, the velocity. Thus, the head losses are the main reason why, any fluid flow which is not produced by gravity, needs a pump. If there isn't a pump, there won't be a flow of water. Indeed, pumps give to the fluid the energy necessary to overcome obstacles, rise through ducts or keep moving despite head losses.

With respect to head losses, they can be separated in two groups depending on what they are caused by:

- First grade losses: it includes all the losses caused by the friction between the flow and the surface of the pipe. They are measured with this expression:

$$h_f = f * \frac{L}{d} * \frac{u^2}{2g}$$

h_f= head losses (m)

L and d= pipe length and diameter (m)

u=flow velocity (m/s)

f= friction Darcy's coefficient

As it can be observed, these first grade losses are influenced by the geometry of the pipes, the fluid flow velocity and the friction Darcy´s coefficient. This coefficient is calculated by using the Moody´s diagram (figure 23) and it depends on the fluid's turbulence (Reynolds number) and the pipe relative roughness, which is quotient between the roughness and the diameter of the pipe.

- Second grade losses: They are also called minor losses or localized head losses and they are related with the obstacles that the flow rate finds in its way through the circuit. Thus, they can be caused by abrupt or smooth contractions or expansions, valves, curves, meshes, pipe divisions etc. All these losses are calculated with the expression below, where the K is a factor experimentally obtained, which measures the impact of each obstacle against the fluid flow:

$$h_v = K * \left(\frac{u^2}{2g}\right)$$

***hv**= localized head losses (m)*

***k**= empiric coefficient*

***v**= velocity (m/s)*

3.2.2. Pipes selection

Regarding the issues that must be considered while designing a pipe system, the most important concerns are head losses (first and second grade losses) and turbulences caused by the pipes. With respect to the head losses, it would be better using big short pipes rather than small long ones. It is because head losses have a directly proportional dependence with the length of the pipes and an inversely proportional one with their diameter. Nevertheless, head losses are influenced, above all, by the velocity of the water, as they are directly proportional to this squared velocity. Taking into account that the velocity of the water depends at the same time on the pipe diameter in an inverse squared way (if the flow rate keeps constant), we can firmly say that the most beneficial pipes are the ones with big diameters.

As far as turbulences are concerned, they depend on the Reynolds number and the obstacles the water comes through. The Reynolds number depends at the same time on the velocity of the water, the diameter of the pipe, and the viscosity of the fluid. With respect to the viscosity, except for changing the temperature, it cannot be changed. Regarding at the other two, theoretically it would be better having bigger diameter pipes, because it would make the Reynolds Number smaller (always talking for a fixed caudal). However, practically, this is not that clear, and the only way I have to try preserving the flow as laminar as possible is trying not to put any object on its way, as it would break the velocity profile, disordering the fluid and provoking turbulences. Besides, turbulences are also brought by any other perturbation in the fluid flow such us convergences, divergences or curves, which I also try to avoid.

As a result from the two studied fields, I reckon that the bigger the pipe is, the less head losses and turbulences are. However, as they are going to be placed in a real half already designed circuit, there are some limits, which in this case are the size and the shape of the tank, and the diameters of the available pipes in the catalogues. Firstly, as the cross section of the flow tank is 20 cm x 20 cm, the diameter of the pipe cannot be bigger than 20 cm. Moreover, the shape of the flow tank obligates to place two 180-degree curves to close the circuit. As I want to do the design as reduced as possible and the pipes as short as possible too, I decide to place them as it can be observed on the figure 16. Secondly and finally, I regard at several enterprises´ PVC pipes catalogues, here in Belgium, and the biggest 90-degree curve pipe which fits in the flow tank and that I have found, has an external diameter of 193 mm and two female extremes. Therefore, I select four of them and a straight pipe that fits in them, whose interior diameter is 144.6 mm. This straight part is going to be divided in three sections, one as long as the flow tank (around 1m), and to small ones (172 mm), which are going to be used for joining two 90-degree curves in order to make the 180-degree one I was talking about. The curvature radius of the curve pipes is 89 mm.

3.2.3. Water Volume

Once the tank and the pipes utilized have been fixed, I can calculate the total amount of water which is going to inside the circuit:

$$V_{Circuit} = V_{Flow\ Tank} + V_{Pipes}$$

On the one hand, knowing that the flow tank cross section is $A = 0.2\ m\ \times 0.204\ m$, and its length is 1 m, I can calculate:

$$V_{Flow\ Tank} = 0.2 * 0.2 * 1 = 0.04\ m^3$$

On the other hand, I can calculate the water volume inside the pipes knowing that the pipe system consist of one 1m straight pipe, two 0,172m straight pipes, and four 90°-degree curve pipes with a curvature radius of 0.089 m; all of them with an inside diameter of 144.6 mm. Hence:

$$V_{Pipes} = V_{Long\ Straight} + 2 * V_{Short\ Straight} + 4 * V_{Curve} =$$

$$= \left(1 * \pi * \frac{0.144^2}{4}\right) + 2 * \left(0.172 * \pi * \frac{0.144^2}{4}\right) + 4 * \left(\frac{1}{2} * \pi * 0.089 * \frac{0.144^2}{4}\right) = 0.025\ m^3$$

Thus, adding the water volume in the pipes and the one in the tank I obtain the total water volume inside the circuit:

$$V_{Circuit} = V_{Flow\ Tank} + V_{Pipes} = 0.04 + 0.025 = 0.065\ m^3$$

4. Water propellers

Firstly, as a clarification, I have to say that this chapter refers to pumps instead of water propellers during almost the whole chapter. This is because, in the beginning, I was looking for a pump, and then, finally, as it was very difficult to find a suitable pump, I decided to use boats propellers. Hence, in this paragraph, I have wanted to tell all the process that has leaded me to the use of boats propellers.

In this design, there are not height differences because the flow moves in an horizontal way, and even if there were height differences, they wouldn't have any importance in a closed circuit, due to the fact that the height, which the water have to climb, makes the same pressure than the pressure made by the water column (owed to this height) in the pump entrance. Therefore, in every close circuit, and especially in this one, the pump is only necessary to make up for the head losses produced by the water during its motion through the circuit. As I have said before, they are divided in two groups, which I will calculate in the next two paragraphs.

4.1. Head losses

As, I said before, head losses are caused by the friction between the flow and the ducts surfaces and they are calculated with this expression:

$$h_f = f * \frac{L}{d} * \frac{u^2}{2g}$$

As my design has different parts with distinct diameters and construction materials, I divide these losses in four parts:

- **Flow tank**: Either the flow tank or the entrance and the exit, both of them are constructed in transparent PVC which has a roughness of ε=0.0000015 m. In order to calculate the friction Darcy's coefficient, it is necessary to know the Reynolds number, the hydraulic diameter of the fluid tank, and its relative roughness. The flow should be as laminar as possible. Yet, it is not easy to make it completely laminar, because of the turbulences made by the pump, the curves etc. Hence, I calculate the Reynolds number, taking as the characteristic length distance from the entrance of the tank. Thus, the Reynolds number is going to increase as the water advance through the tank. In order to make an approximation of the average Reynolds number on the tank, I calculate it in the middle of the tank (x=0.5 m):

$$Re_x = \frac{u * x * \rho}{\mu} = \frac{0.45 * 0.5 * 998}{0.0011} = 204136$$

Moreover, as the cross section of flow thank has a squared shape, we use the expression below to calculate the hydraulic diameter:

$$D_h = \frac{4 * A}{P}$$

D_h= Hydraulic diameter (m)

A= Transversal section (m^2).

P= Wet perimeter (m).

Thus, taking into account the dimensions of the flow rate (0.2 x 0.204 m), and that the upper part of the flow thank is opened, I obtain:

$$D_h = \frac{4 * 0.2 * 0.204}{2 * 0.2 + 0.204} = 0.27 \text{ m}$$

Once I know the hydraulic diameter of the tank, I can calculate the relative roughness of the tank as:

$$Relative\ Roughness = \frac{\varepsilon}{D_H} = \frac{0.0000015}{0.27} = 0.0000055$$

This way, knowing that Re=204136 and ε/D=0.0000055 (smooth pipe), I can check in the Moody's diagram, showed in figure 19, the friction Darcy's coefficient:

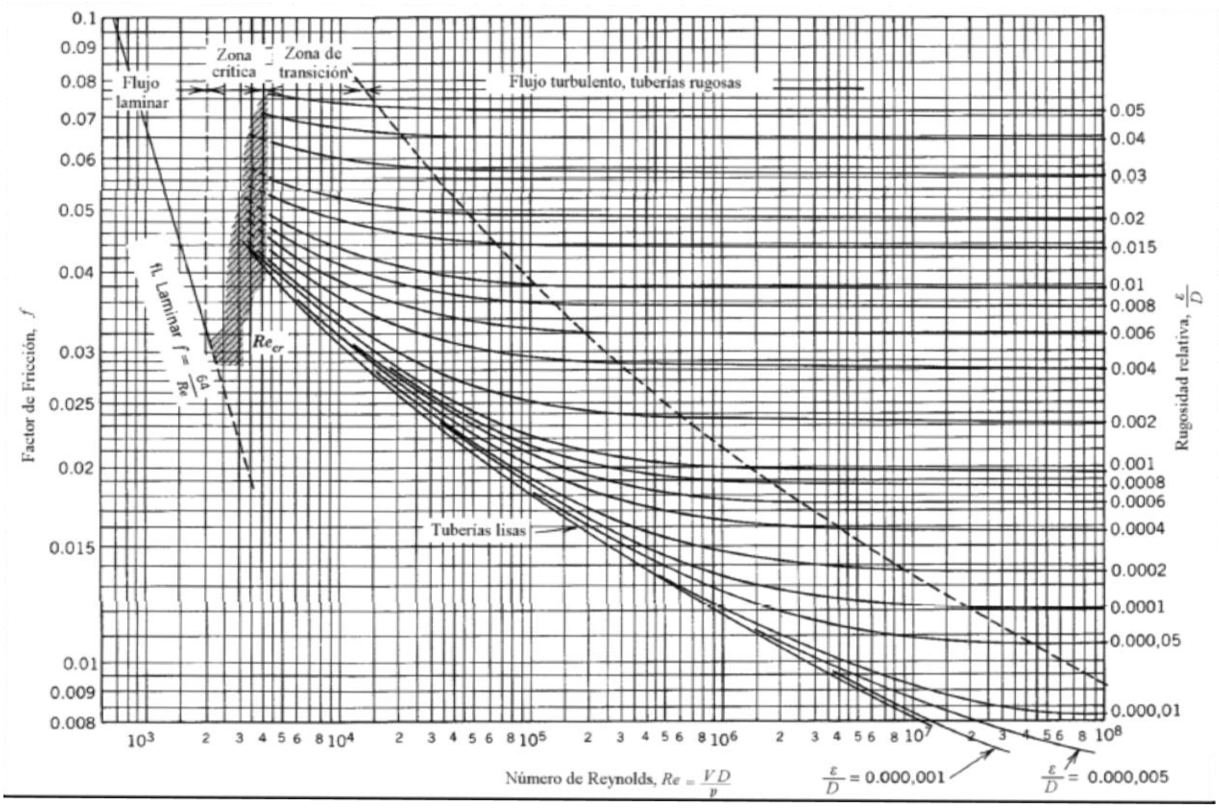


FIGURE19: MOODY'S DIAGRAM.[14]

Thereby, I obtain f=0.016 for calculating the head losses dues to friction in the flow tank. Hence:

$$hf_{Tank} = 0.016 * \frac{1}{0.27} * \frac{{v_{Tank}}^2}{2g} = 0.0030\, {v_{Tank}}^2 \ \text{ m}$$

- **Pipes**: It is important to be noticed that, out of the fluid tank, the fluid flow become more turbulent because of many facts, such as passing through the cylinder (which provokes vortices appearance) or being accelerated through the pipes. That is why I calculated a new Reynolds number. In the case of closed pipes, the characteristic length of the Reynolds number is the pipe diameter. Knowing that the inside diameter of the pipes is 0.1446 m, I calculate the Reynolds number:

$$Re = \frac{u * D * \rho}{\mu} = \frac{0.45 * 0.1446 * 998}{0.0011} = 59036$$

As the pipes are made in PVC too, its roughness is also 0.0000015. Nevertheless, the diameter of the pipes is different from the hydraulic diameter of the cylinder, so I calculate the relative roughness once more:

$$Relative\ Roughness = \frac{\varepsilon}{D_H} = \frac{0.0000015}{0.1446} = \ 0.0000104$$

Then, knowing the Reynolds number and the relative roughness of the pipes, I check in the Moody's diagram that f=0.02 for the rest of our calculations.

It needs to be remarked also that the velocity of the flow in the tank is different to its velocity in the pipes, because of the change of the transversal area and the pressure. Therefore, it is necessary to set an average flow velocity inside the pipes. We are going to figure it out, taking into account that the flow rate is the same in any part of the circuit.

$$A_{Tank} = b * h = 0.204 * 0.2 = 0.0408 \ \text{ m}^2$$

$$A_{Pipes} = \pi * \frac{D^2}{4} = \pi * \frac{0.1446^2}{4} = 0.0164 \ \text{ m}^2$$

$$Q_{Tank} = Q_{Pipes} \ \rightarrow \ A_{Tank} * v_{Tank} = \ A_{Pipes} * v_{Pipes}$$

$$0.0408 * v_{Tank} = \ 0.0164 * v_{Pipes} \ \rightarrow \ v_{Pipes} = 2.545\, v_{Tank}$$

Once the velocity of the flow circulating through the pipes is known, I am going to calculate their utter length in order to be able to calculate the head losses inside them. The total circular pipe length is the addition of the curve pipes length and the length of the straight one:

$$L = \ 2 \ * \ \pi * \ R_C \ + \ L = \ 2 \ * \ \pi * \ 0.089 \ + \ 0.97 + 2 * 0.086 = \ 1.70\, m$$

Knowing that the pipe diameter is 0.2, I calculate the head losses in the pipes:

$$hf_{Tank} = f * \frac{L}{d} * \frac{{v_{Pipes}}^2}{2g} = f * \frac{L}{d} * \frac{(1.27 * \ v_{Tank})^2}{2g} = 2.545^2 * 0.02 * \frac{1.70}{0.1446} * \frac{v^2}{2g} =$$

$$hf_{Tank} = 0.0777\, {v_{Tank}}^2 \ \text{ m}$$

4.2. Minor losses

In this circuit there is going to be also minor losses, which caused the obstacles that the flow rate finds in its way through the circuit. Particularly, due to the geometry of the design, the principal minor losses present on the circuit are those caused by the curves, the meshes, and the abrupt convergence and divergence on the union of the tank with the pipes. All these losses are calculated with the expression below, which have been explained before:

$$h_v = K * \left(\frac{u^2}{2g}\right)$$

Thus, these are the losses that should be considered:

- **Curves**: This design contains four ninety-degree curves. I use the expression below to calculate the empiric coefficient K_C: [15]

$$k_C = 0.051 + 0.19 * \frac{d}{R}$$

Kc=empiric coefficient of the curve
d= pipe diameter (m)
R= curvature radius (m)

Thereby, knowing that our curvature radius is 0.089 m and that the pipe diameter is 0.1446 m, I obtain the secondary head losses due to the curves:

$$Kc = 0.051 + 0.19 * \frac{0.1446}{0.089} = 0.36$$

$$hf_{Curve} = kc * \frac{v_{Pipes}^2}{2g} = 0.36 * \frac{(2.545 * v_{Tank})^2}{(2 * 9.8)} = 2.545^2 * 0.36 * \frac{{v_{Tank}}^2}{(2 * 9.8)} =$$

$$hf_{Curve} = 0.11525\, {v_{Tank}}^2 \text{ m}$$

As there are 4 curves of 90 degrees:

$$hf_{\ Total} = 4 * 0.11896\, v^2 = 0.4758\, v^2 \text{ m}$$

- **Convergence (exit):** The convergence head losses are due to a brusque contraction. The fact that the water flow passes from the squared duct of the flow tank to the circular pipe is associated with secondary losses because of the abrupt change of the cross section diameter of the duct. The coefficient related with these losses can be estimated with the expression below: [15]

$$K_C = 0.42 * \left(1 - \frac{d^2}{D^2}\right)$$

- K_C =Expansion coefficient
- **d & D**=small and big pipe diameter

I calculate the K_C factor:

$$Kc = 0{,}42\left(1 - \frac{0.1446^2}{0.27^2}\right) = 0.3$$

Thus, I can reckon the head losses here as:

$$Hf = kc * \frac{v_{Pipes}^2}{2g} = kc * \frac{(2.545 * v_{Tank})^2}{2g} = 2.545^2 * 0.3 * \frac{{v_{Tank}}^2}{(2 * 9.8)} = 0.0991\, {v_{Tank}}^2 \text{ m}$$

- **Divergence (entrance)**: The divergence head losses are owed to a snappish divergence.The water flow passing from the divergent pipe to the squared duct causes also secondary losses, which can be estimated with a different expression: [15]

$$K_e = \left(1 - \frac{d}{D}\right)^2$$

Ke=*Expansion coefficient*

d & D=*small and big pipe diameter*

I calculate the K_e factor:

$$Ke = \left(1 - \frac{0.1446}{0.27}\right)^2 = 0.215$$

Therefore, I can estimate these head losses as:

$$h_f = kc * \frac{{v_{Pipes}}^2}{2g} = kc * \frac{(2.545 * v_{Tank})^2}{2g} = 2.545^2 * 0.215 * \frac{{v_{Tank}}^2}{(2 * 9.8)} = 0.071\, {v_{Tank}}^2 \text{ m}$$

- **Meshes**: In this experiment, the water flow has to be as laminar as possible, so that vortex shedding could be visualized. As I have calculated before, the Reynolds number of the water flow on the cylinder is 163309. It corresponds to a quite turbulent flow so that meshes/honeycomb structures will be probably necessary. Despite the fact I haven´t put them; I take into account the losses that they can cause, so that if in the end they are used, the pump is powerful enough to compensate their losses too.
  These accessories are made by joining a lot of tiny tubes so that their final looking is similar to those from figure 20:

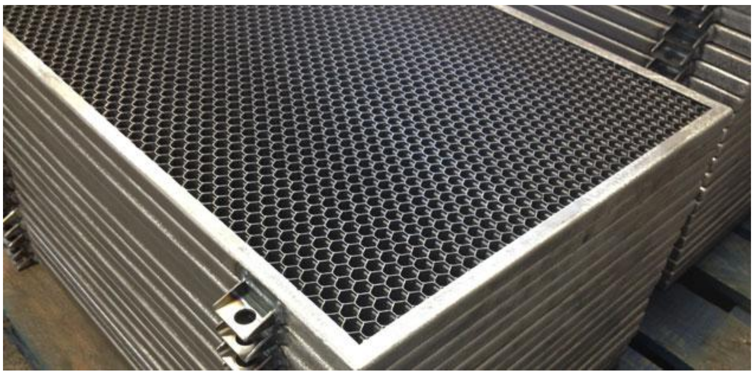


FIGURE 20: MESHES USED FOR MAKING THE FLUID MORE LAMINAR

This kind of structure allows making the fluid more laminar by arranging the flow in small parts. They transform the income flow in many smaller flows orderly disposed.

In order to calculate the head losses through them, it has been necessary to look up on an empiric experiment. There, it was calculated the head losses caused by the water passing through a mesh with a porosity of 40,31% , a crossed section of 1 feet wide and 3 feet deep, and an angle of inclination θ with respect to the free surface of the water flow. In addition, there, it was demonstrated that the head losses due to a water non-Darcy flow (Reynolds' number ≥ 2000) passing though such a mesh are able to be fit by this expression:

$$\Delta H = 132.8 \sin^2\theta \frac{V_1^2}{2g} \qquad \text{when } R_e > 2000$$

where ΔH is in inches and V_1 is in ft/sec. [16]

The Reynolds number of the flow passing through the meshes would be influenced by the distance from the beginning of the tank to the meshes. Therefore, the first one would be the less favorable. Thus, taking into account that the first hole for the meshes is situated at a distance of 0.11 m from the entrance of the tank, I estimate the Reynolds Number of the flow:

$$Re = \frac{u * L * \rho}{\mu} = \frac{0.45 * 0.11 * 998}{0.0011} = 44910$$

As the Reynolds's number bigger than 2000, and assuming that the meshes used have a porosity similar to this experiment's ones, it is possible to calculate the head losses in this design in a sufficiently accurate way. However, it has to be noticed that our meshes would be smaller and they would form a 90-degree angle with the flow. Therefore, a reduction factor is applied to the expression so that losses are proportional to the crossed area. In addition, $\sin\theta$ is one because of the 90-degree angle. But this is not everything; I also have to modify the expression so that it can be used with international system units:

$$1 \text{ inch} = 0{,}0254 \text{ m}, \;\; 1 \text{ ft} = 0{,}3048 \text{ m}$$

$$\Delta H = 132{,}8 * \frac{v^2}{2g} \quad inch \;\rightarrow\; \Delta H = 36{,}308 * \frac{v^2}{2g} \;\; \text{m}$$

$$\Delta H = 36{,}308 * \frac{v^2}{2g} * A_{Design}/A_{\,Experiment} =$$

$$\Delta H = 36{,}308 * \frac{v^2}{2g} * \frac{(0{,}2 * 0{,}2)}{(3 * 0{,}3048 * 1 * 0{,}3048)} = \; 5.24 * \frac{v^2}{2g} \;\; \text{m}$$

Thus, the secondary head losses in the meshes are:

$$\Delta H = 5{,}24 * \frac{{v_{Tank}}^2}{2g} = \; 0.2673\, {v_{Tank}}^2 \;\text{m} \;\rightarrow\; hf \;=\; 2 * \; 0.2673\, {v_{Tank}}^2 = \; 0.535\, {v_{Tank}}^2 \;\text{m}$$

These are all the losses that I take into account. There are more losses such as those which are due to the connection between pipes and the entrance and the exit, the connections with the pump or several valves, which are going to be necessary. Nevertheless, I neglect them, because either they are very small or I cannot calculate them yet, due to the fact that I don't have chosen the pump yet. That is the main reason why I apply a security factor of 1.1 which prevents loads bigger than the estimated ones.

These all losses together, are the head losses of this circuit, and depend only on the installation and on the flow velocity. Thereby, to acquire the pump's power requested by the installation, we only have to add up all the head losses calculated before:

$$H_{Installation} = 1.1 * \sum hf$$

$$H_{Installation} = 1{,}1 * (0.0030 + 0.0777 + 0.4758 + 0.0991 + 0.071 + 0.535) * {v_{Tank}}^2$$

$$H_{Installation} = 1.3877\, {v_{Tank}}^2 \text{ m}$$

4.3. Pump characteristics: Height, Flow rate and Power

Regarding only to the transitory state, the pump has to be big enough to compensate for the head losses along the circuit. Nevertheless, the circuit has firstly a transitory state, in which the pump has to give to the water the energy until that it reaches the stationary state. To calculate the extra power that the pump has to have, I am going to use the conservation

Pumps are usually defined by graphics of height versus flow rate and power versus flow rate. Therefore, I transform this expression in another one which is proportional to the flow rate instead of the velocity, by dividing and multiplying it per the squared crossed section of the flow tank.

$$H_{Installation} = 1.3877\, {v_{Tank}}^2 * \frac{A^2}{A^2} = 1.3877 * \frac{Q^2}{(0.2 * 0.204)^2} = 833.6\, Q^2 \text{ m}$$

Therefore, the head needed by the circuit depends on its squared flow rate, as it can be seen in figure 21. Indeed, for a fixed flow rate, there is a fixed operation point on the curve, and the head of any suitable pump has to intersect with the installation head curve in this point to give the selected flow rate.

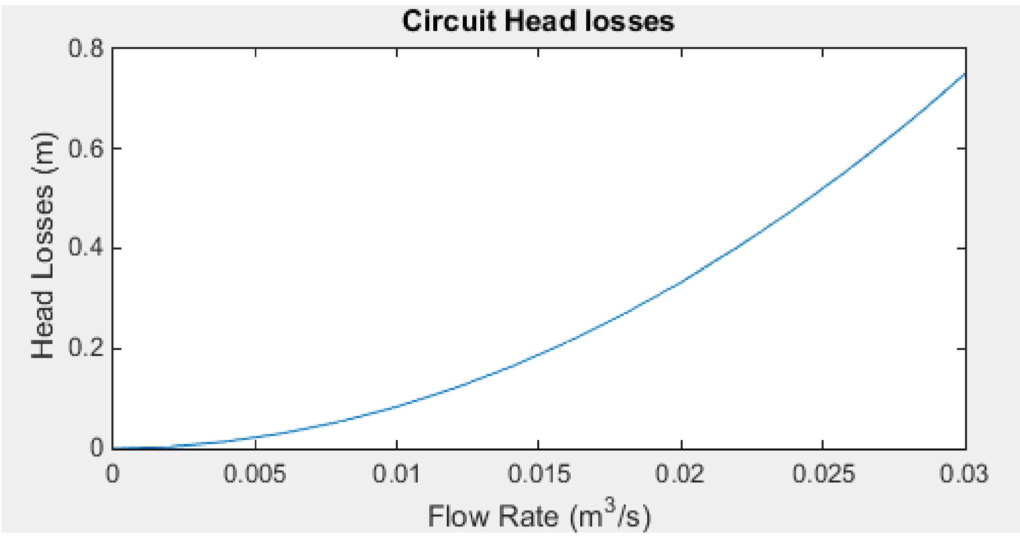


FIGURE 21: INSTALLATION HEAD LOSSES.

In this case, as I have calculated before, the flow velocity on the flow tank is not necessary to be bigger than 0.45 m/s. Hence, the flow rate of the circuit and the pump head can be calculated:

$$v_{Tank} = 0.45 \text{ m/s} \rightarrow \quad Q = v_{Tank} * \text{A}^2 = 0.45 * (0.2 * 0.204) = 0.01836 \text{ m}^3/\text{s}$$

$$H_{Pump} = 833.6 * 0.018^2 = 0.27\, m$$

After calculating the pump´s head, it is necessary to know the pump theoretical power, which is:

$$\mathbf{P = H * g * \rho * Q}$$

P= Pump Power (W)
H=Pump Head (m)
g=gravity $(9.8\, m/s^2)$
ρ=density (kg/m^3)
Q= flow rate (m^3/s)

$$P = 833.6\, Q^2 * g * \rho * Q = 8152941\, Q^3$$

As I have calculated, the power necessary to propel the water flow and surpass the head losses depends on the cube flow rate. This behavior can be seen in figure 22.

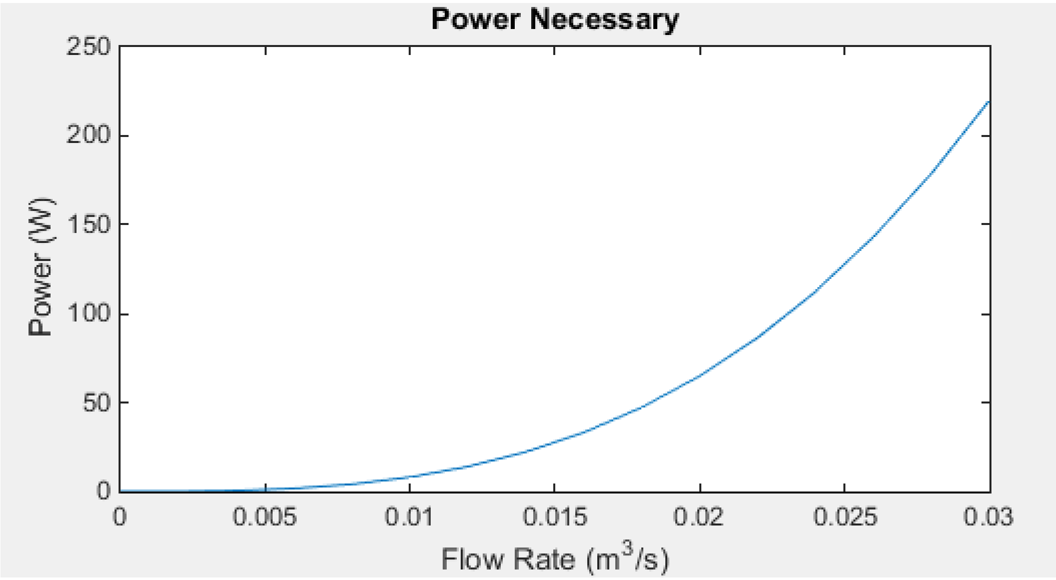


FIGURE 22: NECESSARY PUMP POWER TO PROPEL THE WATER FLOW

In this case, as the flow rate is 0.01836 m^3/s, the minimum theoretical pump power required to move the fluid is:

$$P = 8152941\, Q^3 = 8152941 * 0.01836^3 = 50{,}45\, W$$

Nevertheless, this power only makes reference to the stationary state of the circuit. The actual pump has to give enough power to overcome the head losses and to give the water the kinetic energy necessary to reach the stationary state. Thus, it has to be powerful enough to start moving the water from a state of no motion.

The energy necessary to beat this stationary state can be calculated with the energy conservation equation. However, I don´t calculate its exact value because, as the system size is quite small (compared with huge water installations), beating the transitory state is not going to be an issue. Hence, in order to give enough power to overcome the transitory state without any problem and maintain the stationary state, I raise the needed power to 70 W.

4.4. Selection

Knowing the characteristics that the pump has to have, the next thing I have to do is deciding what type of pipe I choose. In the majority of the circuits, turbo-pumps are selected because they are usually the most useful and cheap pumps. Once one particular pump is chosen and installed in a singular circuit, the flow rate is fixed. This is because of the pump performance, which is going to be explained in two steps. On the one hand, a circuit's head losses depend only on the flow rate (when the transversal area is fixed, on the velocity). On the other hand, pumps can be understood as machines that give an exact amount of energy to each particle the water, so that if the flow rate rises, the amount of water that the pump can give to each particle decreases. Thus, when a pump is placed in a circuit, both the energy that each water particle needs and the energy the pump can give depend on the flow rate, and there is only one flow rate where both of them concur. This point is the working point of the pump, and there, the pump it gives each water particle the energy necessary to stream though the circuit.

Here, the problem is that one of the requirements of this design was the fact that the water flow needs to be able to stream with different velocities. As in most of the cases choosing a pump means fixing the flow rate, it is necessary to vary something in order to accomplish this requirement. There are four possible options to achieve this velocity variation:

- **Two pumps in parallel:** when there are two pumps connected in parallel, the ensemble's characteristic curve is one which, for each height, has a flow rate equal o the addition of the flow rates given by each pump in this height. It is possible to perceive it in figure 23:

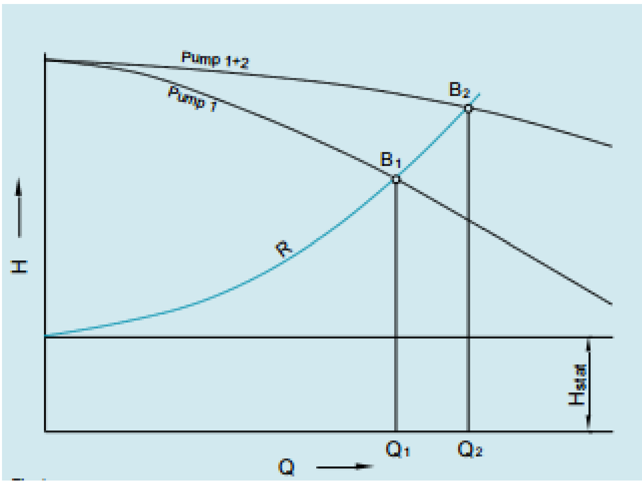


FIGURE 23: PARALLEL PUMPS BEHAVIOR. [17]

- If there are two pumps in parallel, the flow rate rises so that the velocity rises and the head losses increase. Thereby, for the same circuit I can obtain two different velocities. One with both pipes connected, and another one by closing one of the pumps in order to let only the other one work so that the flow velocity decreases. The downside of this method is that this solution is not that easy in the real life because it involves the purchase of two pumps and it complicates the circuit. Besides, it only allows us two different velocities.

- **Velocity variation:** Another possible solution is varying the pump velocity. As the pump rotor velocity decreases, the energy that the pump can give to the water, the flow rate and the power diminish according with resemblance laws:

$$\frac{Q}{Q'}=\left(\frac{n}{n'}\right) \qquad \frac{H}{H'}=\left(\frac{n}{n'}\right)^2 \qquad \frac{Pot}{Pot'}=\left(\frac{n}{n'}\right)^3$$

  This method gives the possibility of several velocities. The rotor pump velocity is the same as the speed of the motor that moves it. Therefore, the number of different velocities that the motor can adopt is the number of different velocities this method offers us. The drawback is that motor speed is determined by the net frequency and the number of pole pairs of the motor, which is not usually higher than four or five. As the net frequency is fixed (50Hz in Europe), the only way to change the velocity of the pump is by changing the number of pole pairs of the motor. This required the using of a special one (winding rotor motor), and it only offers four or five different velocities, as I have said.

- **Variable frequency drive:** Despite the fact that the electric net frequency is fixed, nowadays there exist some appliances that are able to change gradually the frequency of the electricity that supplies a machine. This can allow the pump to have an unlimited number of velocities between two limits that are fixed by the limits of the variable frequency drive bandwidth.(example)

  These appliances do not change the frequency freely. It is necessary that the motor supplied by them maintains the force necessary to move its loads; I mean that it has to maintain its torque. Thus, as the torque is directly proportional to the squared quotient between voltage and frequency, it is enough maintaining this quotient constant:

$$\frac{V}{f}=cte$$

- **Special Pumps:** Finally, as the last option, it is possible to elect directly a pump which allows changing flow rate which stream through it. The problem is that most pumps cannot do this, so it implies buying an especial one such as axial piston pumps which are surely much more expensive than centrifugal ones.

Between these options, I discard using special pipes or special motors, because they are likely difficult to find and expensive. I also discard putting to pumps in parallel because mounting them may be problematic and because they only can allow two different velocities. Therefore, finally, I decide that the easiest and most suitable way of accomplishing our goals is using a turbo-pump with a variable frequency driver, which should be the easiest and more economic solution, as it is the most common combination. With respect to the pump, there are three groups of turbo-pumps depending on their specific velocity: centrifugal ones, diagonal ones and axial ones. As it can be observed in figure 24, the centrifuge pumps are suitable when propelling low flow rates, needing high heads, whereas the axial ones are adequate when needing large flow rates to be propelled little heads. Therefore, it seems that this project needs an axial pump due to the small head and the circuit range of flow, but I will calculate it afterwards.

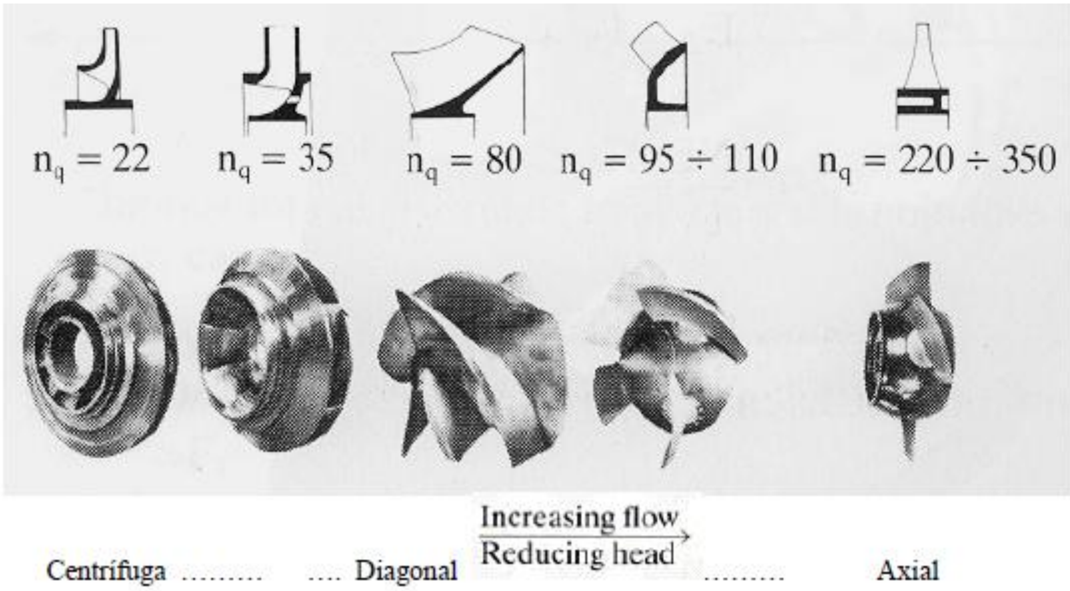


FIGURE 24: DIFFERENT KINDS OF TURBO-PUMPS DEPENDING ON THE SPECIFIC VELOCITY.[18]

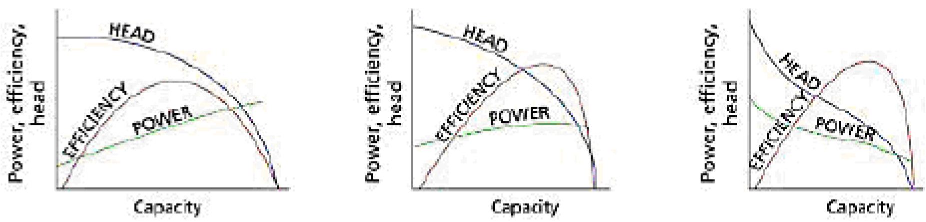


FIGURE 25: CENTRIFUGE, DIAGONAL AND AXIAL PUMPS BEHAVIOR.[18]

The specific velocity is a parameter that relates the installation requirements with the kind of turbo-pump that it needs. Hence, in order to select the most appropriate pump for the project, I calculate the specific velocity that I need according to my previous calculations: [18]

$$n_q \equiv n_{sp} = \frac{n * \sqrt{Q}}{H^{3/4}}$$

n_q , n_{sp}= Specific Velocity

N= Pump angular velocity (rpm)

Q= Flow rate (m^3/s)

H= Head on the operating point (m)

Thus, assuming a velocity of 1500 rpm, I obtain a high specific velocity: $n_q = 537.28$, as can be watched bellow. It means that I need a pump completely axial.[18]

$$n_q = \frac{1500 * \sqrt{0.018}}{0.27^{\frac{3}{4}}} = 537.28$$

Once I have arrived at this point, I look for a suitable pump for this circuit, but I find no pump totally adequate. Indeed, the only pumps that operate giving this low head either are prepared to move much bigger quantities of water, so that they don´t fit in my design, or they are working really far from their design point, and so, their efficiencies are pretty bad (50%-60%). That is because the operation point I need is quite bizarre: the specific velocity is very high, which means that the suitable pump has to be very axial. In other words, the flow rate is very big in relation to the necessary head.

Nevertheless, despite the fact it is really difficult to get a suitable pump, a great solution has been found; I am going to use boat propellers supplied, as they are really similar to axial pumps. Indeed, they are machines which are able to move large amounts of water at low velocities, and this is what I need. In addition, as they are simple elements (helix), they can be purchased in almost every size, what means that they allow a simple and adaptable way of being mounted, giving flexibility to the design. Furthermore, they not only offer us a comfortable and accurate way to move the water flow, but also permit us to reach a much more affordable solution for the problem, owe to the fact that I am changing a ten thousand-euro pump for a fifty-euro propeller.

4.5. Propellers Structure

Once I have selected boat propellers to move the water flow, I have to decide how many of them I am going to use, how big they are going to be, and how I am going to place them in the circuit.

Regarding at its placement, it is needed to be noticed that the propellers produce turbulences in the water. Therefore, ideally, they must be placed inside the pipes, right after the cylinder, so that the turbulences they make don´t affect the vortex shedding and, accordingly, the results of the practical experiment. Nonetheless, placing them inside the pipes brings several problems. Firstly, as the pipes are closed ducts, it would be necessary to put both the propellers and the motors that move them inside the water, which would be quite complicated due to the fact that usual electronics cannot be wet. Secondly, it has to be taken into account that the pipes are glued between them, and so, everything inside them is inaccessible. Thus, putting them inside the pipes would mean that if something fails, it could not be repaired. And finally, the last reason to avoid this ideal solution is that there is not enough physical space inside the pipes to put more than a simple propeller in the same cross section, as the internal diameter of the pipe is 144.6 mm. According to all of these reasons, there is no other way than placing the propellers inside the flow tank, even if this fact brings turbulences near the cylinder. In order to minimize the turbulences, the propellers are located at the end of the tank, after the cylinder, so that most of the turbulences they produce go downstream, directly to the pipes. Yet, the effect of the propellers is tangible even before the water reaches them. In fact, the presence of a propeller spinning in the water provokes a vortex starting some distance before it. That is the main reason why the propellers are place as far as possible of the cylinder, trying to prevent these turbulences interfering in the experiments results.

As I have said, the propellers have to be as far of the cylinder as possible. Therefore, they are going to be place all in the same cross section. Considering that the cross section of the flow tank is squared, there are only two possible configurations that propel the water homogenously: one big propeller on the centre of the cross section, or four small propellers in each corner of the cross section. Nevertheless, the first one possibility is discarded because it will introduce a big rotational moment due to its spinning. In addition, the size of the propeller and the size of the motor driving it are related, and as the motors I am going to use are small, using four small propellers is more adequate than using only a big one.

The diameter of the propellers is limited by the cross flow section too. As it measures 0.2m x 0.204 m, I use four diameter 75 mm boat propellers to impulse the water though the circuit.

Regarding at the moment of rotation in the water, it has to be noticed that the spinning of each small propellers is going to produce a small local moment of rotation too. However, the global moment of rotation can be annulated by selecting the right configuration of the propellers. Thus, in order to minimize the moment of rotation, A-B-A-B is the best propellers configuration. With this configuration, two diagonal propellers rotate clockwise whereas the other two spin counterclockwise, as it can be seen in figure 26.

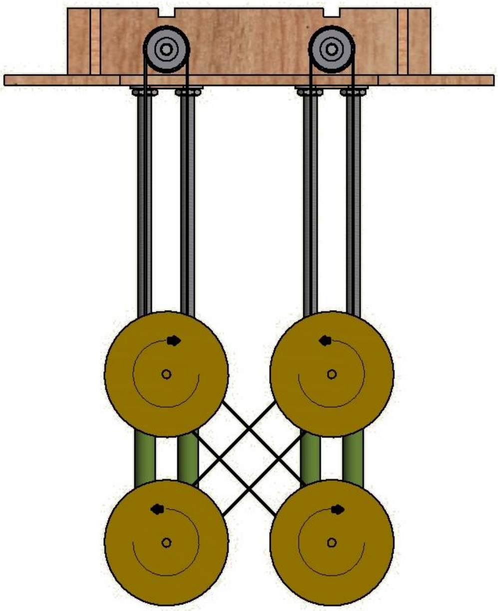


FIGURE 26: PROPELLERS A-B-A-B DISTRIBUTION.

In addition, with respect to the propellers positioning, a supporting structure needs to built, because they cannot support by themselves. As a consequence, I design a structure which allows placing them inside the water and transmitting the motors movement to them. Nevertheless, it is not enough holding the propellers in any way; it is necessary a lightweight hydrodynamic structure, because large pieces inside the tank would obstruct the flow coming though the propellers, and abrupt structures would cause a lot of turbulences, which is not good neither for the experiments nor for the propellers, as they function better with laminar flows. As a result, I design a fluid dynamic structure that supports the propellers, which can be observed in picture 27. It is composed of the next elements:

- Basis Plate: This plate is screwed on the top of the frame surrounding the tank. With other little plates help, it supports two motors, each one supplying current for two propellers. Furthermore, it also holds 8mm eight threaded bars which hold the rest of the structure.

- Threaded bars: As I have said, there are eight, and they are though as the skeleton of the structure. They are as slim as possible as possible to prevent the obstruction of the water flow thought the propellers. Taking into account that rigidity is also necessary in the structure, I select 8mm columns

- Bullets: These pieces are the key of the design. There are four of them in the design, each of them containing inside them two bearings, which are tightly gripped by the bullet, and a propeller axis, which can rotate inside the bearings. The most important of these pieces is that their shape, similar of a bullet, makes the whole design fluid dynamic. Moreover, they have been made in a way that they are as small as possible, reducing enormously the size of the structure. These last two things allow the flow passing the structure almost without obstruction, which is really important for the correct performance of the propellers.
  The bullets are wanted to be placed and fixed in the correct place so that the propellers are inside the water and don´t hit between themselves or to the tank walls. In order to fix the relative position between the bullets, the upper propellers are separated from the lower ones by hollow columns that are introduced in the threaded bars between the bullets. Besides, several nuts are utilized to fix their absolute position.

- Axis and Propellers: As I have said before, there are four 75mm diameter propellers in this design. Each of them is threaded to an axis so that both of them move together. These axes have to be supported in two points by two bearings that allow them to rotate smoothly. There are two circlips, one in each side of one bearing, that prevent the axis to move forwards or backwards.

  Timing belts: In addition, as far as the propellers movement is concerned, they have to be connected with the motors because they are the ones that cause the rotational movement. To solve it, a transmission system based on belts and pulleys is placed between the axes and the motors, permitting the transmission of the motion. This transmission system utilizes timing belts because they ensure that the motors velocity and the propellers velocity are going to be the same (in the case of using same size pulleys for the axes and the motors). In order to accomplish the rotation configuration explained before, each timing belt is going to join two diagonal axes, as it can be observed in figure 27:

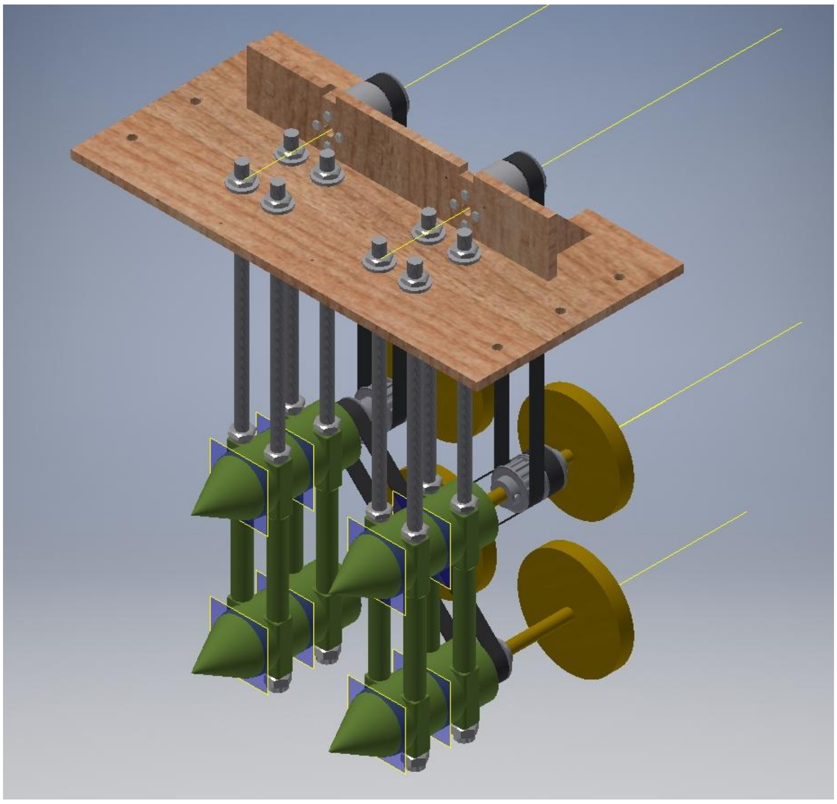


FIGURE 27: PROPELLER STRUCTURE.

4.6. Cavitation

Cavitation is a problem that can appear in every machine related with fluid motion. It can be due to two factors, both of them related with the low pressure on the entrance of the pump:

- Air dissolved in water: Water has always some air dissolved inside it. This air can evaporate on the entrance of the pump due to its low pressure.
- Water state changes: These state changes can be produced at any temperature depending on the pressure. Water evaporates and form bubbles when the pressure is lower to the vapor pressure at the temperature of the water, as it can be beheld in figure 28.

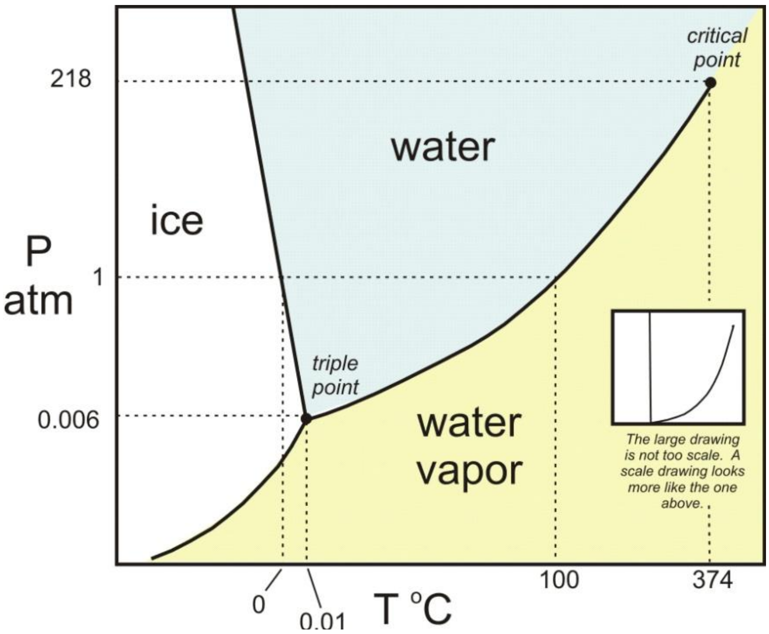


FIGURE 28: WATER PRESSURE-TEMPERATURE DIAGRAM.

Thus, if the pressure in the entrance of the propeller is low enough, water evaporates and there appear vapor bubbles which are in equilibrium with the water and follow the watercourse downstream. The issue is that, when the pressure increases, the vapor becomes water once more and the bubble collapses. An implosion is produced which generates a pressure peak. This pressure peak is really dangerous because it can damage the propeller and the ducts walls, as we can see in the picture 29.

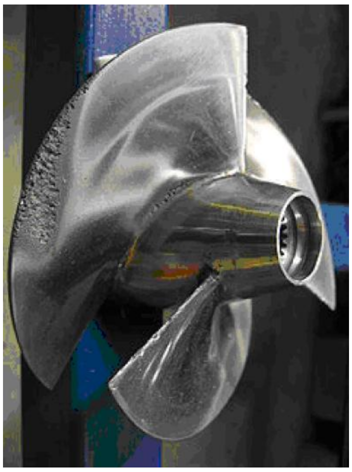


FIGURE 29: HELIX DAMAGED BY CAVITATION.

There are two parameters intended to control and prevent this phenomenon to happen. These two parameters are: NPSH available (Net Positive Suction Height), which refers to the maximum head losses that can be in before the pump so that cavitation doesn´t appear; and NPSH required, which are the head losses produced inevitably by the propeller. In order to prevent cavitation, NPSH available has to be bigger than NPSH required. Yet, it is usually used a security factor:

$$NPSHrequired = NPSHavailable + 0.5 \text{ m}$$

The NPSH required, which only depends on the installed pump, is almost always given by the suppliers, whereas the NPSH available only depends on the installation and have to be calculated. Hence, I am going to reckon it:

$$NPSH\ available = \frac{P_e}{\gamma} + \frac{{v_e}^2}{2g} - \frac{P_v}{\gamma}$$

The Bernoulli equation says that, when there are not head losses, total energy in the water get conserved:

$$\frac{P}{\gamma} + \frac{v^2}{2g} + z = cte$$

This equation can be evaluated between the entrance of the propeller and one of the points of the water surface of the flow tank, taking into account that there are going to be head losses as it is a real circuit. Considering the height of the water surface as zero:

$$\frac{P_{atm}}{\gamma} + \frac{{v_{Tank}}^2}{2g} = \frac{P_e}{\gamma} + \frac{{v_e}^2}{2g} + z + hf_{1\to e}$$

$$\frac{P_e}{\gamma} + \frac{{v_e}^2}{2g} = \frac{P_{atm}}{\gamma} + \frac{{v_{Tank}}^2}{2g} - z - hf_{1\to e}$$

If this last expression is substituted in the NPSH available equation, I obtain:

$$NPSH\ available = \frac{P_{atm}}{\gamma} + \frac{v_{Tank}{}^2}{2g} - z - hf_{1\to e} - \frac{P_v}{\gamma}$$

This formula is the most commonly utilized one to calculate the NPSH available. It usually depends only on the installation characteristics, because normally, the Bernoulli equation is evaluated in a point of still water. However, here, the NPSH available depends also in the propellers because of the way of evaluating the Bernoulli equation in a moving point, which introduces the factor $\frac{v_{Tank}{}^2}{2g}$.

Yet, to continue with the calculus, I can´t take this value as the most unfavorable one, which is the lowest velocity the water is supposed to move, v= 0.15 m/s. It can be estimated that there is going to be a high value of NPSH available as the subtracting values are very small. Assuming a water temperature of 20 degrees, it can be looked up in tables that the vapor pressure of the water at this temperature is 0,023393 atm, which is quite low. Moreover, the difference of heights is going to be negative, as the center of the propeller is going to be submerged (I omit it). In addition, the head losses of this part of the circuit are going to be lower that the head losses of the whole circuit, so they are going to be really low (I take them as they were the losses of the complete circuit. Therefore:

$$NPSH\ available = \frac{P_{atm}}{\gamma} + \frac{v_{Tank}{}^2}{2g} - z - hf_{1\to e} - \frac{P_v}{\gamma} = \frac{100000}{9800} + \frac{0.15^2}{2 * 9.8} - 0{,}63 - \frac{2339}{9800} = 9.34\ m$$

It results a NPSH available of 9.34 meters. This is a very high NPSH and it is going to be always bigger than the NPSH required by any pump. Therefore, I have proved that there is not going to be cavitation in this design.

5. Frame

The frame is the part of the design in charge of supporting the rest of the design and the loads acting in the flow tank. As it can be observed in figure 15, it consists of a structure constructed with 3 x 3 mm aluminium profiles, which are joined between them using triangular pieces and screws. It is necessary to notice how important this element is, as it permits the performance of the rest of the design, allowing placing and connecting each part of the design in the right way. Thus, its main two functions are supporting the pipes so that they are place at the same height than the holes of the flow tank caps, and bearing the loads due to pressure coming from the flow tank.

The first task is achieved by using five 8mm burch plates that are going to support the pipes maintaining them at the same height than the flow tank, making possible a perfect watertight connection between them. In this case, the plates are screwed to the aluminium structure. However, there are not necessary many screws in each plate owe to the fact that the plates are supported on the floor. Hence, there are not going to be the screws but the floor which makes the force to support the weight of the pipes, being the screws useful for joining the plates to the structure and fixing their position.

In order to accomplish its second task, a box is made surrounding the flow tank. It is composed for 8 vertical bars in each of its sides, joined in their lower and upper extremes with bars passing from one side to the other side above and beneath the flow tank. This box alleviates the efforts that the joins of the PVC plates have to support, and it prevents the large transparent PVC plates from bending because of the water pressure. Regarding at this purpose, it can be notice that the aluminium profiles are not working at all, as this kind of structure is thought to work in contraction and not in traction, and these loads only make the upper bars work in traction. Therefore, the screws that join them are the pieces that support all the efforts due to the water pressure. Nevertheless, although it could be consider as a waste, these profiles and this structure has been selected because it is the most suitable way of achieving these two aims in an easy and satisfactory way.

6. Driving mechanism

As said before, this project has to contain an oscillating cylinder. As it isn't going to move by itself, it is necessary to build a driving mechanism which allows the cylinder to do the required movement.

But what is exactly this movement? As it is said in the requirements, it is binding that the cylinder oscillates somehow in a plane perpendicular to the water flow. However, I have not been said how this movement has to be. Thus, the first dilemma appears here, because there are two possibilities: on the one hand, I can position the cylinder in a horizontal way and force it to move vertically, and on the other hand, I can place the cylinder vertically and configure the mechanism so that the cylinder moves horizontally.

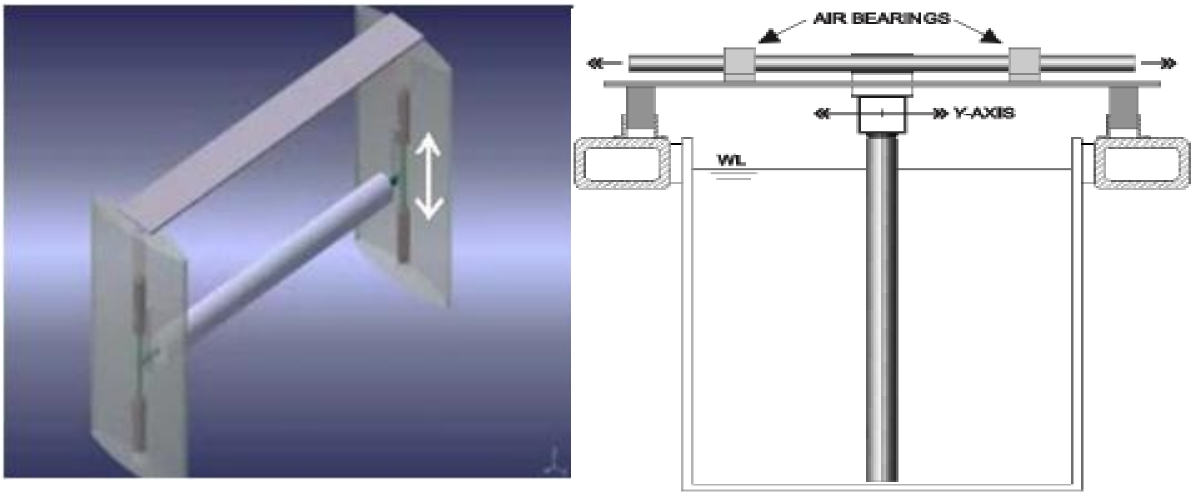


FIGURE 31: A HORIZONTAL CYLINDER MOVING VERTICALLY (ON THE LEFT) AND A VERTICAL ONE MOVING HORIZONTALLY (ON THE RIGHT).

Which option is better? In order to evaluate this, I must bear in mind several aspects. Firstly, I invite you to observe that in the first option the whole cylinder is inside the water. The electronics of the mechanism has to be as far as possible from the water to keep it dry. Hence, this fact doesn't help, because if I used this way of motion, maybe I would need waterproof electronics, which is much more expensive. On the other hand, it is easily seen that one the vertical cylinder extremes is never wet; so looking at this criteria, it seems better the second possibility. Besides, it also can be observed that the cylinder placed horizontally needs two gripping points whereas the other cylinder has only one. Thus, it would be more complicated joining the mechanism with the cylinder in the first option than in the second. And finally, it is also necessary consider that, in the first movement, gravity would be actuating, as in every vertical motion, while in the vertical cylinder gravity can be negligible, as it doesn't act in the direction of the movement. Therefore, I can conclude saying that I choose the second king of motion because it would be much more difficult to accomplish the driving mechanism by using the first kind of movement.

Thereby, the cylinder is placed vertically and it moves horizontally along a plane perpendicular to the water flow. Nonetheless, the cylinder movement is not completely defined yet. There are still few aspects to take into account in order to set the motion of the cylinder:

- Function: the cylinder could move along the plane perpendicular to the water flow following infinite kind of movements. Hence, it is necessary to select one function that describes the motion of the cylinder. In this design, this function is going to be sinusoidal, as I have been said that there is not necessary a more complicated function. Thus, the cylinder does a periodic movement between two positions, which are not set yet.

- Stroke: This is the parameter which fixed the two points that the cylinder moves between. This parameter is important only for the visualization of the vortices. In fact, the distance of the path has to allow a clear vortex shedding. For this objective, it is sufficient with a distance of three times the diameter of the cylinder. As the diameter of the cylinder used is 30 mm, the path has to measure something around 90 mm. Thus, respecting this, I fix a distance of 100 mm for the cylinder motion. However, although it is not necessary, it would be great if this distance could vary a little bit, so that experimenters could perform different proofs, obtain different results and compare them. Hence, even if the main distance is 100 cm, the crank is going to be designed in a way that allows different stroke possibilities, as it is going to be described below.

- Velocity: Once the function and the distance have been fixed, there is only the velocity which lasts unfixed and it is important to establish it because it is directly related with the vortex shedding. After the cylinder's lock-in, both the vortex shedding frequency and the cylinder oscillations frequency fall together, which mean that the quicker the cylinder moves, the quicker the vortices are shed. Thus, the velocity has to take a value that allows the vortices being shed in observable frequencies.

  As I have said during the design of the flow tank, it would be great having different frequency possibilities for the vortex shedding. In order to do it, I need a motor which is able to change its velocity. The velocities that it has to be able to give are restricted by the vortex shedding frequencies that allow us to watch clearly the vortices, so that each shedding frequency is related with a cylinder velocity and a rotor velocity. It is possible to observe it in the table 3:

Frequency (Hz)	Velocity Cylinder (m/s)	Rotor Velocity (rpm)
1	0,15	60
1,5	0,225	90
2	0,3	120
2,5	0,375	150
3	0,45	180

TABLE 3: MOTOR ANGULAR VELOCITY RELATED DO THE CYLINDER FREQUENCY

It can be observed that we want the cylinder moving at the same velocity than the water flow. It is because we are looking for the lock-in phenomenon.

6.1. Rod Mechanism

Normally, the design and construction of a driving mechanism depends, above all, on the motion that the driven objet is going to describe. There exist a lot of kinds of mechanisms that allow designers to achieve almost every movement they want. The most common mechanisms are based on elements such as rod mechanisms, gears, pulleys and cams. Each one permit doing a large range of movements, and sometimes two of them, or even more, can be used to accomplish the same motion. Therefore, the election usually depends on their set up easiness.

The motion that the cylinder has to follow is a sinusoidal function. As it is a quite easy function, I can achieve it by using a simply rod mechanism. Once it is known that a rod mechanism is going to be used, it is necessary to start thinking about where it is going to be placed, how many bars it is going to have, or how they are going to be joined. The easiest mechanism that I can use is a rod mechanism consisting of two bars and a guide, where the shortest bar makes complete circles while the other one rotates a determinate angle, as it can be observed in figure 32. Respecting to the guide, it is only a hole that permits the cylinder moving straight.

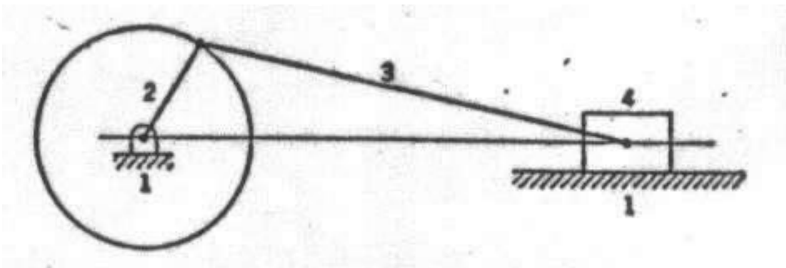


FIGURE 32: STANDARD ROD MECHANISM. IT IS EASY OBSERVE THAT THE BAR 2 MAKES COMPLETE CIRCLES WHILE THE BAR 3 ONLY ROTATES A DETERMINATE ANGLE, AND THE CYLINDER (NUMBER 4) FOLLOW A PERIODIC MOVEMENT THROUGH A STRAIGHT PATH.

In regard to the setting place, the most comfortable way of collocate the driving mechanism is by building a plate above the flow tank and putting the mechanism there. It would be also possible placing it beside the flow tank, but it has the drawback that it would be necessary to build another structure to bear it. As the construction of this structure would be much more difficult than building a simple plate, we are going to select the option of putting the mechanism above the tank. In addition, it is quite easy designing a guide in each side so that the plate is able to slide along the tank. This can permit the cylinder modifying its position, and going a little bit upstream or downstream.

Thus the whole mechanism, which can be observed in figure 34, can be explained in the next few points, in spite of the fact that more pieces are going to be required:

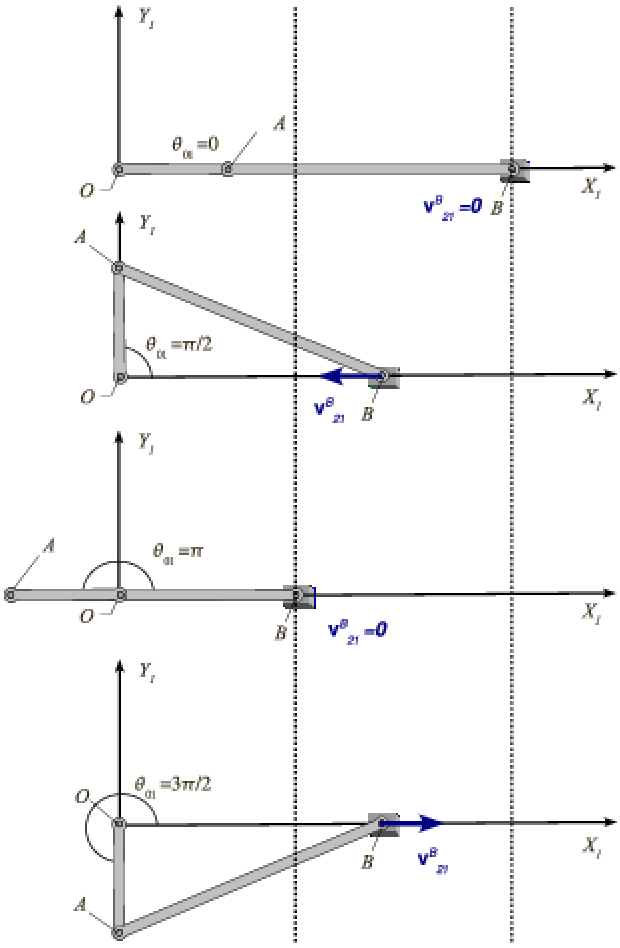


FIGURE 33: FOUR MAIN POSITIONS OF THE ROD MECHANISM USED IN THE DESIGN.

- Motor: A motor is absolutely necessary in every driving mechanism because it is the piece that transforms the energy in movement and introduces the motion into the mechanism. There are different kinds of motors but I am going to use an electric one, because they are the cheapest and most efficient motors. As the movement issued from the motor is a rotational movement, the motor has to be joined directly with the first bar, which is going to make circles.

- Crank: It is a circular plate which possesses several holes, each one situated at a different distance from the axis of rotation. It is connected with the motor and makes a circular trajectory. As I have said before, it is commonly called crank, and it is responsible for the periodic movement of the cylinder. This periodicity is due to the circular trajectory, and the relationship between both of these movements can be explained in one period, as we can observe in figure 33: at the beginning of the circular path, the cylinder is situated in one of the sides of the guide. When the crank starts making the circle, the cylinder set off its path towards the other side. It arrives there when the crank has travelled half of the circle, and there, the cylinder velocity is zero. As soon as the crank continues walking the second half of the circle, the cylinder begins coming back to its initial position. It arrives there when the crank has drawn a complete circle, and in this position, the cylinder velocity is zero once more. Hence, it can be perceived that one period of the crank motion is related with one period of the cylinders movement.

- Connecting rod: This piece consists of a narrow plate with two holes, one in each extreme of the plate, as can be observed in the planes attached to the document. It doesn´t have any special function, but it is necessary for the transmission of the movement. It has one of its ends bonded with the crank and the other one with the cylinder, and its motion only covers a determinate angle and not a whole circle.

  Once both of the pieces have been explained, it is needed to notice that the relationship between the bars lengths is one of the most important issues of this kind of mechanism because it fixes the size of the mechanism, the angle that the connecting rod is going to travel and the effect that the geometry causes on the forces transmitted through the mechanism. Between these two measures, the crank one is the most significant due to the fact that its length determines the stroke, or the distance that the cylinder travels. Paying attention in picture 33, it can be observed that the length of the cylinder´s stroke is twice the length of the crank. Therefore, as the distance has to measure around 100 cm, the crank has to measure 5 cm. Nevertheless, as I have said before, it would be right letting the distance of the cylinder path change. Thus, the crank consists of a circular piece with several holes, each of them placed at a different distance from the center of rotation. They allow many different distances for the cylinder stroke, between mm 50 and 120 mm, depending on which hole is joined with the connecting rod. On the contrary, the length of the connecting rod is not important at all. In fact, there are only two physical limits that the design has to fulfill. Firstly, the length of the connecting bar has to be at least twice longer that the other one´s length, because if not, it is physically impossible that both extremes of the bar are joined at the same time with the crank and the guide respectively, during many times of the motion. Yet, it is not only sufficient this first requirement, because even if the connecting bar is twice bigger than the crank, it is possible that some pieces hit between them because of its own thickness. Thus, secondly, this bar has to be big enough to prevent any hit between any parts of the mechanism. Therefore, considering both of these requirements, I set the length of the connecting rod in 15 cm, as a design decision.

- Besides, it is also important the position of the center of rotation of the first rod. Any angle (distinct to zero) formed between the center of rotation and the line pointed by the guide would trigger a quick return movement. As the cylinder is required to perform a sinusoidal movement and I want it to travel in both senses with the same velocity, the center of rotation of the crank has to be in line with the guide, so that there is not any quick return.

- Guide: Usually, motion gives by motors is a rotational movement. Thus, every time we want something to do a linear movement, it is necessary to place a guide that only permits this kind of movement. In this case, as the cylinder has to perform a linear path, there is a rectangular hole and a guide on the plate that allow the cylinder moving forward and backwards. The dimensions of the hole are: 5 cm of width, so that several cylinders with different dimensions fit correctly inside it, without touching the edges of the hole, which would bring undesirable friction; and 154 cm of length, so that it can travel its maximum stroke without any trouble.

- Cylinder: The cylinder is the most important piece of the mechanism, as it is the obstacle that produces the vortex shedding. This design is made in a way that allow the experimenters to change the diameter of the cylinder whenever they want (logically, the diameter of the cylinder has to be always smaller than the width of the hole). This diameter is related with the vortex shedding frequency through the Strouhal equation. Thus, if a big diameter is selected, the water flow velocity has to be bigger to achieve the same vortex shedding frequency. As a consequence, I select a diameter as small as possible, taking into account that the cylinder has to be big enough to produce well visible vortices. Hence, initially, I set its diameter in 3. cm. Regarding at its length, it is a more complicated issue because it depends on the thickness of the limit layer. Basically, the cylinder is wanted to intercept always water streaming with the same velocity. Therefore, the cylinder mustn´t be in contact with the limit layer, because it would confuse the experiments results, as the fluid particles in the limit layer are influenced by the viscosity and its velocity is smaller. In order to know the maximum length that the cylinder can measure, I estimate the thickness of the boundary layer. For laminar boundary layers over a flat plate, the Blasius solution to the flow gives:

$$\delta = 5 * x/\sqrt{Re_x}$$ δ=Boundary layer thickness (m)

Knowing that x=0.4m and Re_x= 163309, as I have calculated before:

$$\delta = 5 * \frac{x}{\sqrt{Re_x}} = 5 * \frac{0.4}{\sqrt{163309}} = 0.00495 \text{ m } \rightarrow 4.95 \text{ mm}$$

Hence, theoretically, the cylinder could be immersed until 19.505 cm. However, as it is not necessary to submerge it that much, I submerge it 15 cm in the water.

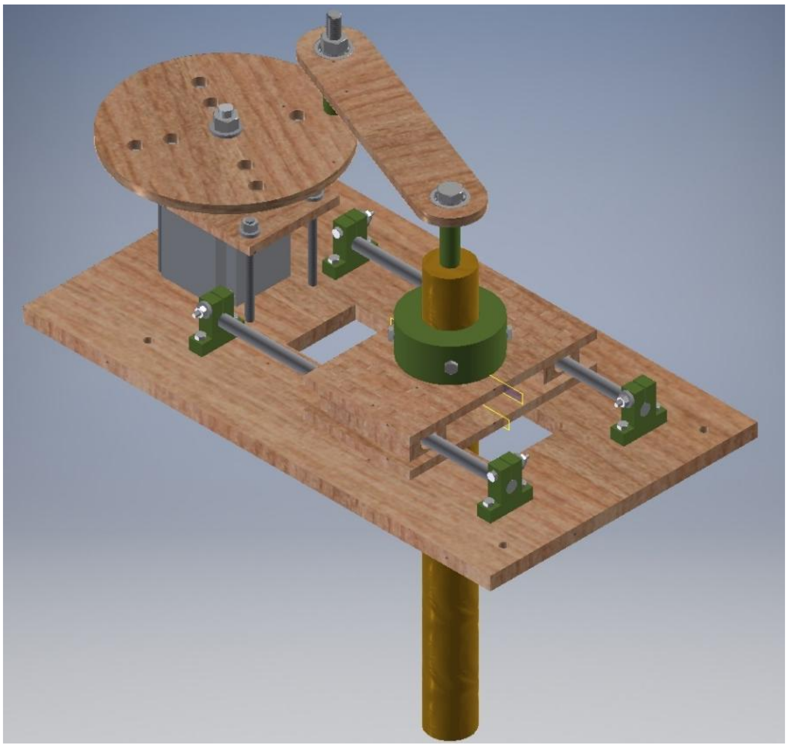


FIGURE 34: DRIVING MECHANISM.

6.2. Connections

One of the most important parts in a mechanism are the connections between its bars. Defining well the permissions and restrictions on each connection is essential for the correct mechanism performance. Thus, these are the present connections in the mechanism:

Motor and crank: Both of these pieces are charged with the transmission of the movement towards the mechanism. The motor rotating part is a not threaded cylinder. Thus, I can join it directly to the crank. That is why I utilize a plastic piece called to transmit the movement to the crank. It as a hole where I introduce the motor rotating part, and I tighten it with two small screws. This piece has a protrusion with a characteristic shape which fits exactly in a hole of the crank which has the same shape. Thank to this, both components make the same movement.

- Crank and Connecting rod: One of the ends of the connecting rod moves along the guide while the crank makes circles bonded with the motor. As a consequence of this, the direction of the two rods is changing and the angle between both of them is not constant. Therefore, it is necessary placing between them a piece that permits them rotating without friction. This piece is a bearing, which forbids all of the possible motions between the objects unless rotation.

- It would be difficult, if not impossible, joining the crank and the connecting rod only with a bearing, due to their shapes. Accordingly, I use a screw which is only threaded on the end of its length and a hollow column whose function is separating the connecting rod from the crank. Thus, the screw is introduced on the hole of the bearing of the connecting rod, then the column is introduced in it, and finally it is introduced through the hole of the crank and fixed with a washer and a nut. Regarding at the union between the outer race of the bearing and the connecting rod, the tightening of the join is going to accomplish by making the diameter of the connecting rod hole a little bit smaller.

- Connecting rod and cylinder: It is easy to observe that the cylinder slides along the guide without rotating whereas the connecting rod changes its direction during its motion. Hence, once more, it is going to be necessary a bearing between these two pieces, that allows a rotation between them without friction. Although both of these two pieces could be joined directly to the bearing, I use another piece to bond the cylinder with the bearing, as in the previous union. Indeed, I utilize the same screw and the same column that I used before, but instead of tighten the screw with a nut, I tighten it directly to the cylinder. Thus, by using this piece, I allow an easy change of the cylinder, as the only required thing is a cylinder with a screwed hole on its top. Therefore, this design permits the experimenters to attach there almost any cylinder they are interested in.

- Cylinder and Guide: As I have said before, the cylinder slips along the guide. This kind of movement is always related with a high value of friction. However, in order to minimize it, linear bearings are placed in two axes situated in both sides of the cylinder, being able to move along them. Despite the fact that is seems that only two linear bearings are required, I am going to use four, because using only two to could lead to a misalignment between the bearing and the shaft, which would prevent it sliding correctly. In order to join the cylinder with the bearings, I design a kind of box that frames the linear bearings and which has two plates on its tow and its bottom with a hole where the cylinder fits in. I have deciding putting two plates instead of one, because having two contact points prevent the cylinder from bending due to the water forces. Besides, it can be noticed that the cylinder is not leant on anything, which means that its weight would push it down if I did nothing. The impact of this would be a disaster, because it wouldn´t only damage itself, but it also would cause the bending of the connecting rod and even the crank. This is a big issue owing to the fact that the connecting rod is long and slim, so that even a small force in one of its extremes may cause a large flexion moment that breaks the piece. Thus, in order to solve this problem, I design a piece with a ring shape. This piece, which is support in the box superior plate, is going to surround the cylinder and grip it tightly with four screws. This way, the weight of the cylinder is transmitted thought the screws to the ring and supported by the box plate.

6.3. Forces acting on the cylinder. Motor Torque

Once the mechanism is selected and all the measures are fixed, it is necessary to calculate the torque and the power required to carry out the wished movement.

The torque is the ability of forces to provoke a rotational movement in an object. It is also, commonly called, moment of forces, and it is always referred to a center of rotation. Its value depends on the direction and the magnitude of the forces acting on the object and their distance to the center of rotation. Accordingly, it is easily perceived that the torque not only depends on the forces but also on the geometry of the mechanism. It is calculated with the following equation:

$$\vec{\tau} = \vec{F}\, x\, \vec{r}$$

T= motor torque (N*m)
F= Tangential force (N)
r= Distance from the force to the center of rotation (m)

There are several forces acting on this driving mechanism but not all of them are significant. In fact, some of them are negligible because of different reasons. In this case, all of the existing forces are acting on the cylinder, but not all of them have influence on the motor. Thus, the forces acting on the cylinder are:

- Gravity: This force acts in every object existing in the earth. Nevertheless, it is not always important, as in this mechanism. The main reason why it is not important here, is that it is not acting in the plane of rotation and therefore, it doesn´t have any influence on the motor. All its action is borne by the plate where the mechanism is placed, and consequently, by the frame.
- Friction: The friction appears during the movement of the cylinder along the guide. There are rotational ball bearings and linear bearings helping to the cylinder motion so that, theoretically, the friction force should be small. That is why, it can be negligible.
- Inertia: The cylinder is a body which is continually accelerated and decelerated in its path. Accordingly, there is and inertia force doe to these acceleration changes which can be calculated with the Newton second law.
  In addition, there exists another inertia force, which is the inertia of the motor. It is due to the acceleration of the motor and it only affects to the mechanism during the transitory state because during the transitory the angular acceleration of the motor is 0.
- Drag force: These two last forces are fluid dynamic forces caused by the interaction between the fluid and the cylinder. As I have said before, the drag force is a sinusoidal force that always pushes the cylinder towards the direction of the relative fluid flow. In this case, despite the fact that the absolute water flow velocity is parallel to the tank walls, the drag force is not pointing exactly at this direction. This is because this force, as the lift force, depends on the relative velocity between the water and the cylinder, as it can be seen on the figure 35. Hence, it can be seen that the drag force is parallel to this relative velocity and the lift one perpendicular to it. Therefore, drag can be decomposed in two components, one parallel to the water flow and another one perpendicular to it.

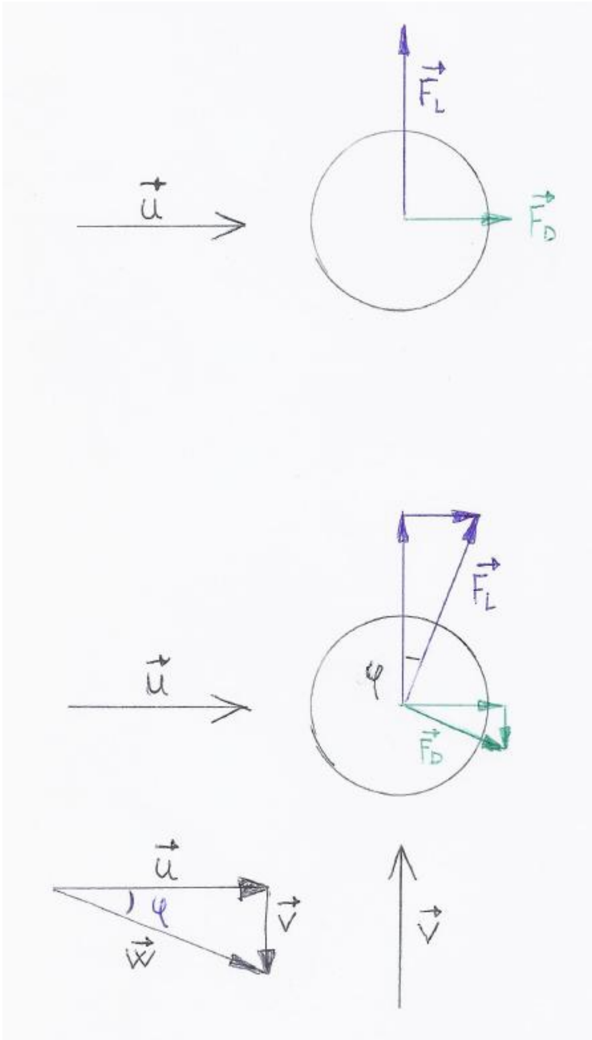


FIGURE 35: FLUID DYNAMIC FORCES ON A STILL (UP) AND MOVING CYLINDER (DOWN)

On the one hand, regarding at the parallel component, it doesn´t have any effect on the motor because the mechanism basis plate absorbs all its actions, as it only permits the movement of the cylinder in a direction perpendicular to the water flow.

On the other hand, the component perpendicular to the water flow does affect the motor torque, as the cylinder is free to move in this direction so that the effect of this force is transmitted through the bar mechanism until the motor.

- Lift force: This is the other force caused by the vortex shedding. Alike to the drag force, the direction of this force depends on the relative velocity of the water from the cylinder. However, this force acts perpendicularly to this velocity, following also a sinusoidal behavior and pushing the cylinder to both senses on the cross flow direction. Therefore, this force has also two components, a small one parallel to the flow, which doesn´t affect the motor torque as it is supported by the mechanism basis plate; and another bigger one perpendicular to the fluid flow, whose action is free to move the cylinder. Thus, this component is suffered by the whole mechanism and supported by the motor.

6.3.1. Geometry

Nonetheless, before starting discussing the influence of the forces on the motor, I am going to analyze the mechanism geometry, which introduces some limitations in the mechanism, and modifies the forces effects on the motor.

If we looked the driving mechanism from its top, its geometry can be trivialized to a triangle, as we can observe in figure 36. Looking at the figure, I can define the angle β as the angle formed between the line that joins the center of the motor and the connecting rod, and the horizontal axis of the figure. In the same way, the angle α can be defined as the angle formed by the connecting rod and the horizontal axis.

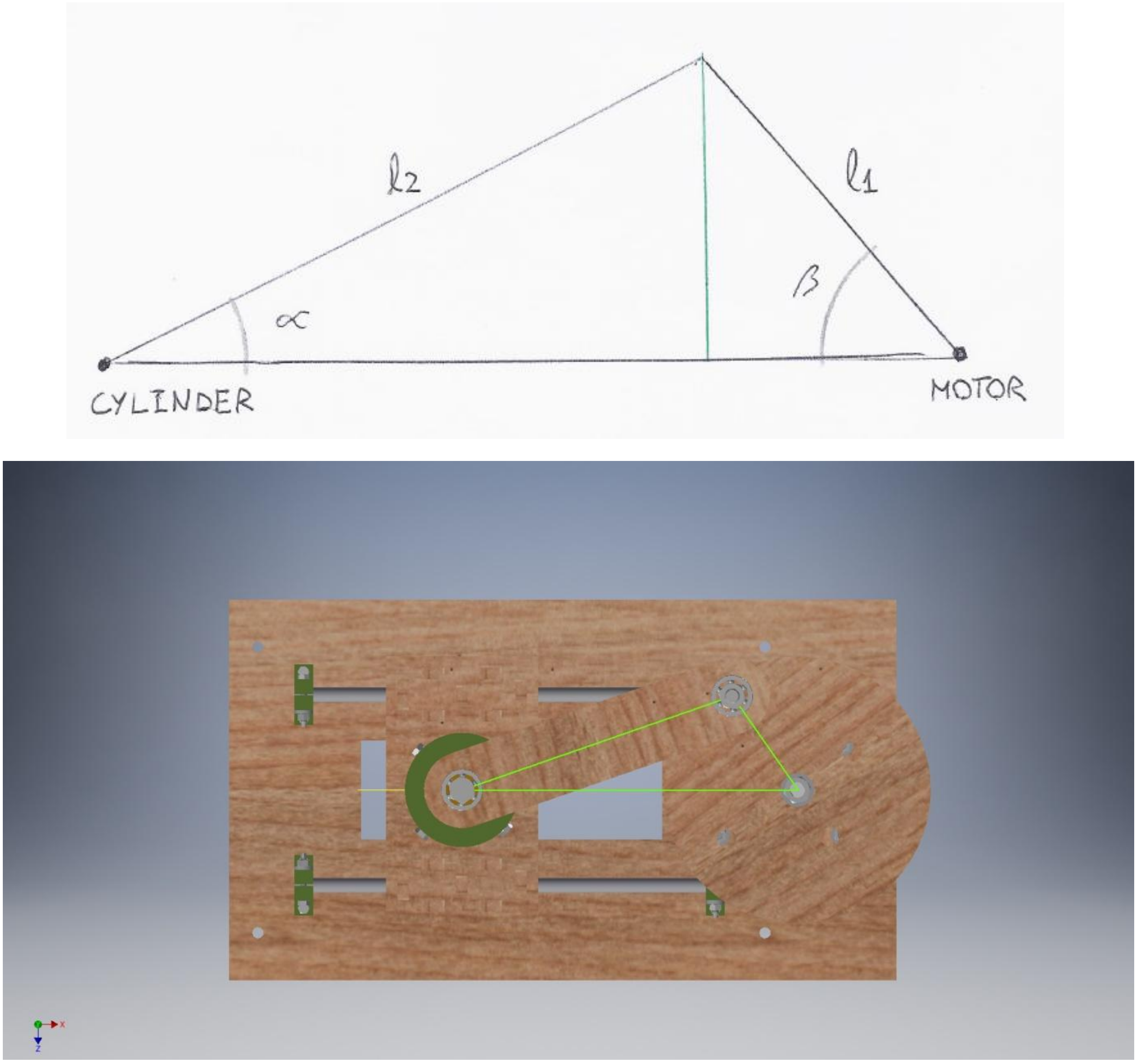


FIGURE 36: GEOMETRY OF THE DRIVING MECHANISM

It is not complicated to observe in the figure 36 that there is a restriction in the mechanism. The heights of one of the end points of the crank and the connection rod have to be the same. Thus, if I take l_1 as the length of the crank, and l_2 as the length of the connecting rod, I obtain the following constraint:

$$l_1 * \sin\beta = l_2 * \sin\alpha$$

$$0.05\ m * \sin\beta = 0.15\ m * \sin\alpha$$

$$\frac{\sin\beta}{\sin\alpha} = 3$$

In addition, due to the way that the rods are connected, it is possible to observe that the angle α adopts its biggest value when the angle β is 90 degrees. Hence, I calculate the maximum value of the angle α:

$$\begin{cases} \frac{\sin\beta}{\sin\alpha} = 3 \\ \beta = 90° \end{cases} \rightarrow \alpha = [19.47°, -19.47°]$$

6.3.2. Cylinder movement, velocity and acceleration

As it has been said before, the cylinder forced oscillations are sinusoidal and their frequency is the frequency of the motor because of the direct transmission of the bar mechanism. Thus, the frequency is changeable, as well as the amplitude, due to the several possible positions of the crank. Nevertheless, in order to do this calculation, I am going to fix them, so that the frequency is going to be 1 Hz and the stroke of the cylinder 0.1m. Thus, setting the origin on the center of the stroke and assuming that the starting position of the cylinder is going to be in one of its ends, I can adjust the movement of the cylinder during the stationary state to the function below, whose behavior can be observed in the figure 37:

$$y = A * \cos\beta = 0.05 * \cos\beta$$

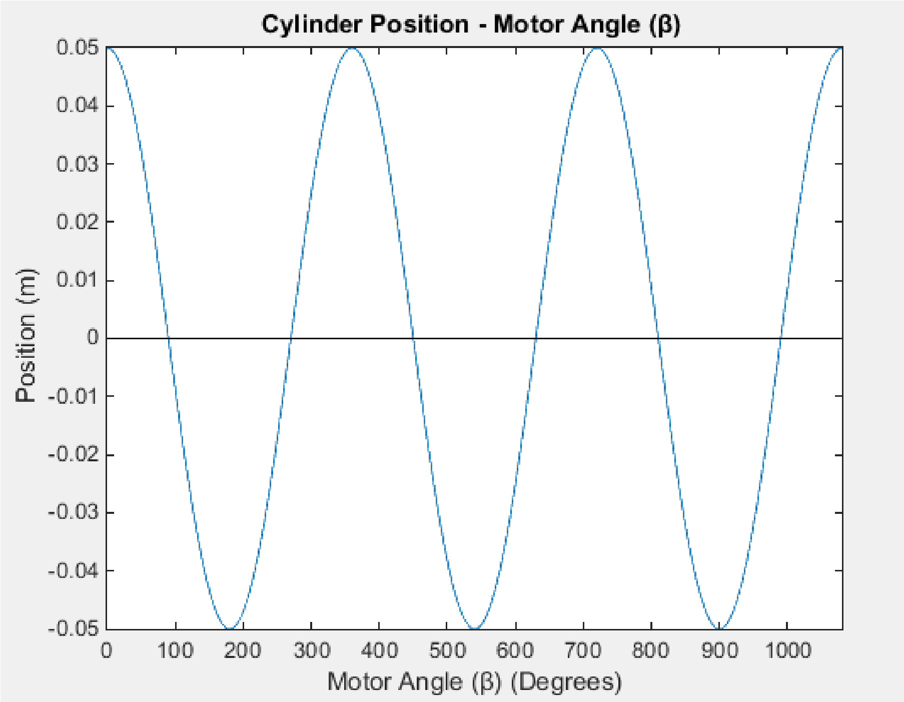


FIGURE 37: CYLINDER MOTION.

As a mathematic concept, velocity is the derived of movement with respect to time. However, in this case, the cylinder movement doesn't depend on time, but on the motor's angle. Therefore, I calculate the velocity of the cylinder during the stationary state. Below this assumption, the motor angular velocity is constant, and therefore:

$$f_C = 1Hz \rightarrow \omega = \frac{2\pi}{1} = 2\pi \text{ rad/s}$$

$$v = \frac{dy}{dt} = \frac{dy}{d\beta} * \frac{d\beta}{dt} = \frac{d\,(0.05 * \cos\beta)}{d\beta} * \omega = -0.05 * 2 * \pi * \sin\beta = -0.314 * \sin\beta$$

The behavior of the velocity can be observed in figure 38:

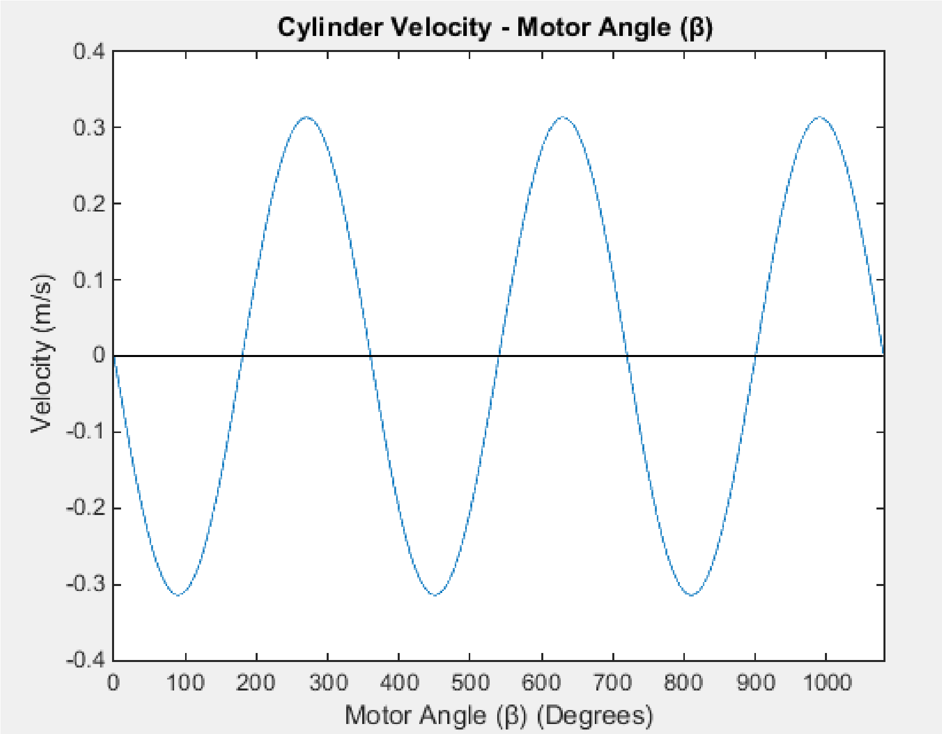


FIGURE 38: CYLINDER VELOCITY.

Regarding at the acceleration, as a mathematic concept, it is the derived of movement with respect to time. Below the assumption of stationary state, the motor angular velocity is constant, and I can calculate the acceleration of the cylinder as:

$$a = \frac{dv}{dt} = \frac{dv}{d\beta} * \frac{d\beta}{dt} = \frac{d\,(-0.314 * \sin\beta)}{d\beta} * \omega = -0.314 * 2 * \pi * \sin\beta = -1.973 * \cos\beta \ \ \mathrm{m/s_2}$$

The behavior of the cylinder acceleration can be observed in figure 39:

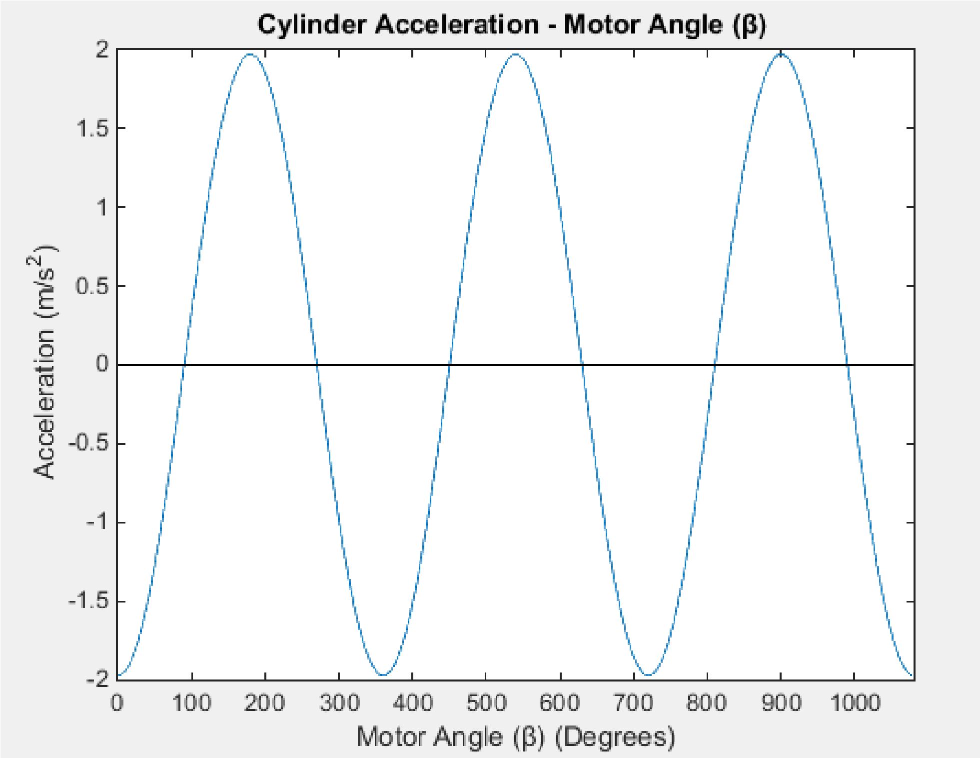


FIGURE 39: CYLINDER ACCELERATION.

6.3.3. Lift Force

In this chapter, I am going to try to adjust the lift force to a mathematical expression. As it has been said before, this force is a pulsating force. Indeed, it varies between a maximum value and a minimum one, following the sinusoidal behavior of its lift coefficient, C_L, which can be seen in figure 14. This C_L behavior, and accordingly the lift force, they are related with the shedding frequency and therefore, they have nothing to do with the cylinder frequency during non lock-in conditions.

Thus, this force depends on the water currents and the vortex shedding mode so that is quite difficult to model it. However, in order to do the calculations, I suppose that the velocity of the water is 0.45 m/s. So, according to Table 2, vortex shedding frequency is 3 Hz, three times more than the assumed motor frequency. To calculate the amplitude of the lift force, it is necessary to calculate before the lift coefficient, which depends on the Reynolds number of the incoming flow. Therefore, I evaluated the Reynolds on the cylinder for the assumed flow velocity (u=0.45 m/s), knowing that the characteristic length is the cylinder diameter:

$$Re = \frac{u * D * \rho}{\mu} = \frac{0.45 * 0.03 * 998}{0.0011} = 12250$$

As I have said before, the lift coefficient is a sinusoidal function so that its average value is cero. However, the sectional root-mean-square lift coefficient can be estimated using experimental equations which fit its behavior. Accordingly, as the calculated Reynolds number is 12250, I utilize this equation: [19]

$$For\ 5400 < Re < 220000 \rightarrow C_L{}' = 0.52 - 0.06 * \left(\log \frac{Re}{1600}\right)^{-2.6}$$

$$C_L' = 0.52 - 0.06 * \left(\log\frac{12250}{1600}\right)^{-2.6} = 0.4373$$

The root-mean-square of a sinusoidal wave is its amplitude divided by $\sqrt{2}$. Thus, I can calculate the amplitude of the lift coefficient, and express it as a sinusoidal wave:

$$C_L' = \frac{A_{C_L}}{\sqrt{2}} \rightarrow A_{C_L} = \sqrt{2} * C_L' = \sqrt{2} * 0.4373 = 0.6184 \rightarrow C_L = 0.6184 * \cos\theta$$

It can be established a ratio between the assumed 3-hertz vortex shedding frequency and the 1-hertz cylinder frequency (motor frequency). Hence, below these assumptions, the lift coefficient can be expressed as a sinusoidal wave depending on the motor angle β:

$$\begin{cases} C_L = 0.* \cos\theta \\ \cos\theta = \cos 3\beta \end{cases} \rightarrow C_L = 0.6184 * \cos 3\beta$$

Thus, the lift force is modeled with this equation:

$$\begin{cases} F_L = \frac{1}{2} * C_L * \rho * u^2 * S \\ C_L = 0.6184 * \cos 3\beta \end{cases} \rightarrow F_L = \frac{1}{2} * 0.6184 * \rho * u^2 * S * \cos 3\beta$$

Finally, I obtain the mathematic expression of the lift force, whose behavior can be observed in figure 41:

$$F_L = \frac{1}{2} * 0.6184 * 998 * 0.45^2 * 0.03 * \cos 3\beta = 1.8746 \ \cos 3\beta \quad \text{N per m of immersed cylinder}$$

6.3.4. Drag Force

To adjust the drag force to a mathematic equation, firstly, I have to study the behavior of its coefficient C_D. As I said before, the drag coefficient consist of two parts, a constant one, C_{D_o}, and a oscillating one, C_{D_I}, which depends on the lift force. I study each of these coefficients separately:

- C_{D_o}: Parasitic drag only depends on the shape of the body obstructing the fluid flow and the Reynolds number. Therefore, I can obtain its value from the graphic of figure 40.

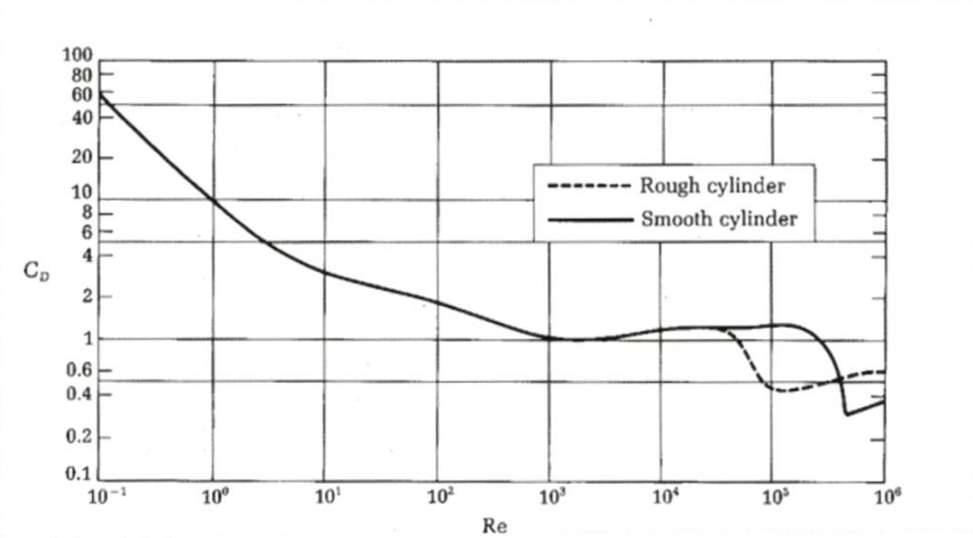


FIGURE 40: PARASITIC DRAG COEFFICIENT VERSUS REYNOLDS NUMBER (CHARACTERISTIC LENGTH = CYLINDER DIAMETER)

Knowing that the Reynolds number on the cylinder is 12250, I obtain from the graphic the value of the drag parasitic coefficient: approximately, C_{D_o}=1.2.

- C_{D_I}: Induce drag depends on the lift force perpendicular to it. Thus, the C_{D_I} of the component of the drag force parallel to the cylinder movement is only influenced by the component of the lift force parallel to the fluid flow. Therefore:

$$C_{D_I} = \frac{(C_L * \sin\varphi)^2}{\pi * e * AR} = \frac{(C_L * \sin\varphi)^2}{\pi * e * \frac{L}{D}}$$

The Oswald efficiency number is always below 1, and it usually takes values between 0.6 and 0.99. Thus, I assume $e = 0.75$, as a middle value. With respect to the aspect ratio, in the case of a cylinder it is the ratio between its immersed length and its diameter. In this design, the immerged length is 10 cm and the diameter of the cylinder is 3 cm, as I have said before. As a consequence, AR= 10/3=3.33. Now, knowing these two parameters, I can calculate the C_{D_I}:

$$C_{D_I} = \frac{(C_L * \sin)^2}{\pi * e * AR} = \frac{(C_L * \sin)^2}{\pi * e * AR}$$

$$C_{D_I} = \frac{(C_L * \sin\varphi)^2}{\pi * e * AR} = \frac{(0.6184 * \cos 3\beta \; * \sin\varphi)^2}{\pi * 0.75 * 3.33} = 0.048 * \sin\varphi^2 * \cos 3\beta^2$$

It can be observed that the amplitude of this very small compared with the drag parasitic coefficient or the lift coefficient. That is the main reason why I neglect it.

Accordingly, neglecting the induced drag, the drag coefficient is equal to the parasitic drag coefficient and therefore, it is constant.

$$C_D = \ C_{D_o} = 1.2$$

Hence, the modulus of the drag force is calculated utilizing the equation below, being v the cylinder velocity:

$$F_D = \frac{1}{2} * C_D * \rho * v^2 * S = F_L = \frac{1}{2} * 1.2 * 998 * 0.314^2 * 0.03 = 1.771\ N \text{ per m of immersed cylinder}$$

This drag force component it is directly proportional to the squared velocity, as it can be observed in the previous equation. Regarding at its direction, it is always against the movement of the cylinder (against its velocity) so that it can be considered as a friction force. Therefore, the drag component in the cylinder direction can be modeled like this:

$$F_D = \ 1.771 * \sin\beta \text{ N per m of immersed cylinder}$$

The behavior of this force can be observed in figure 41:

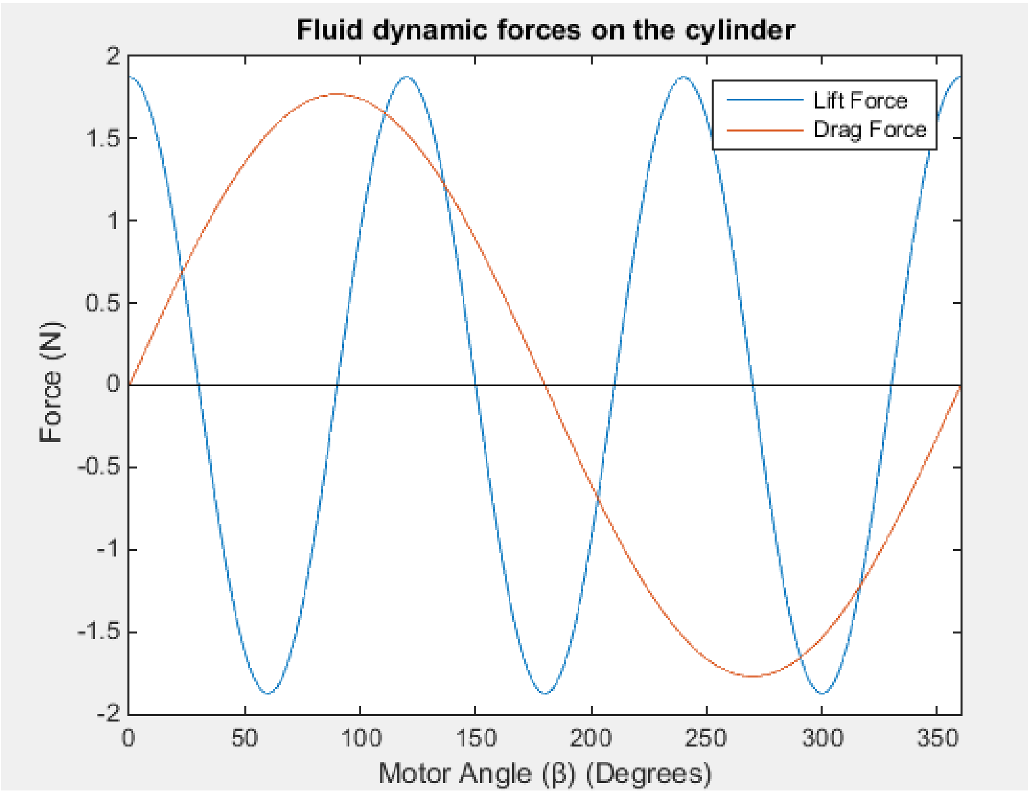


FIGURE 41: LIFT AND DRAG FORCES.

6.3.5. Trigonometric factor

Once both geometry of the mechanism and the forces action on it have been analyzed, I can focus on how these forces are going to affect to the motor torque. Firstly, it is necessary to know that, when a moment of forces is calculated, there are two different kinds of forces:

- Forces pointing towards the center of rotation: this kind of force doesn´t have any influence at all in the torque of the motor, because it cannot provoke a rotation or a change on the circular movement of an object around this center of rotation.
- Forces perpendicular to the radius of a center of rotation: these are the forces that generate moment of forces because all of their action is aimed to revolve (around an axis) the objet that they are pushing.

Any other force acting in any other direction has to be decomposed in forces acting in the directions mentioned before, introducing a trigonometric factor in the solution. This way, it is easy to observe how much influence a twisted force has on the moment of forces.

In this case, the axis of rotation is the center of the motor, and the forces acting freely on the cylinder that can affect to the rotation of the motor, are lift, drag, and inertia forces. The drag force acts always against the cylinder movement whereas the lift one sometimes helps the cylinder motion and sometimes obstructs it. However, these forces don´t have a direct effect on the torque of the motor, because of the fact that they are not acting directly on it, but they are transferred until the motor by a complete rod mechanism. This transfer has, as a result, forces with different directions acting on the motor which should be decomposed in radial and tangential forces. Consequently, the total influence of lift, drag and inertia forces on the motor is modified by a trigonometric factor that is going to be calculated right now.

In order to make this calculation, I suppose that the motor rotates anti-clockwise and I only analyze the effect of the drag force. As it can be consider as a special friction force, I draw it against the cylinder motion. I use a random position of the mechanism to determine the trigonometric factor that relates this force with its final tangential component acting on the motor. For realizing this study, I decompose the drag force in two components which I analyze in two different figures (42 and 43): In the first one I break down the component of the force on the direction of the connecting rod, while on the second one, I dissect the component of the drag force perpendicular to the connecting rod:

- Component parallel to the connecting rod: this component is transmitted until the extreme of the crank. There, this force can be decomposed in a component pointing towards the center of the motor, which doesn´t create torque, and another one tangential to it, creating torque on the motor.

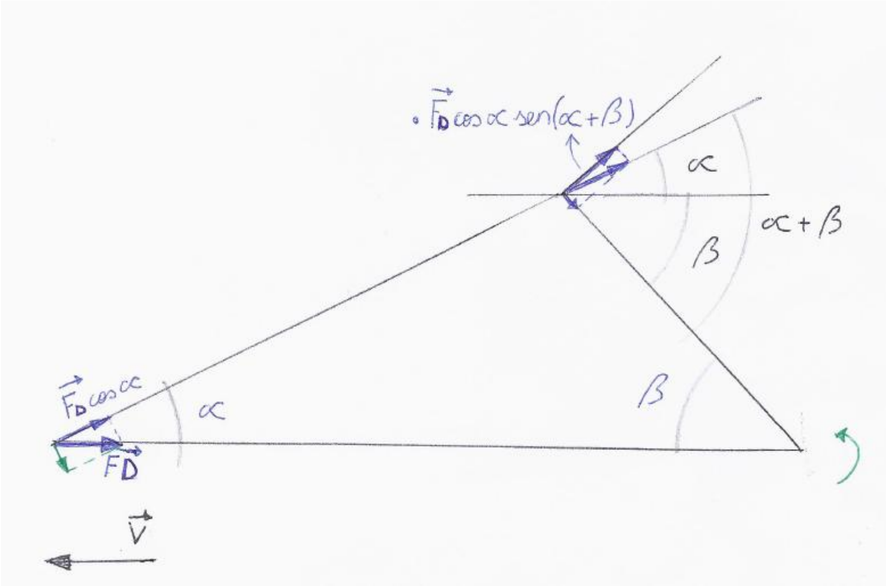


FIGURE 42: EFFECT OF THE DRAG FORCE COMPONENT PARALLEL TO THE CONNECTING ROD ON THE MOTOR TORQUE.

In figure 42, it is developed the effect on the motor of the drag force component parallel to the connecting rod. How does the transmission works? This component obligates the connecting rod to do a compression work so that it transmits the equal force to the crank. As these bars have different directions, it is necessary to make another decomposition. As a result, this force is divided in two new components; one parallel to the crank and pointing to the motor axis of rotation, and another perpendicular to the crank, tangential to the movement. As I have said before, only the perpendicular forces have an effect on the torque of the axis of rotation. According to it, I am only interested in the tangential component.

Thus, in figure 42 it is possible to see that the drag force component parallel to the connecting rod has a negative contribution on the moment of forces, as the final tangential force is acting against the rotation. Therefore, it tries to brake the motor. Furthermore, reflecting about the drawing, it can be generalized that, at any angle of the circle, this force is acting against the motor. Indeed, between 0° ≤ β ≤ 180° this force is always trying to rotate the crank clockwise. For 180° ≤ β ≤ 360°, it has to be heeded that, as the cylinder is moving towards the other way, the direction of the drag force changes so that the it continues acting against the motor in this half of the circle.

As the resulting tangential force is going to be always applied in the same sense of the direction perpendicular to the crank, I am going to consider, in advance, this sense of forces as the positive one. Hence, finally, the force influencing the moment of forces due to this component is:

$$F_{Parallel} = F_D * \cos\alpha * \sin(\alpha + \beta)$$

- Component perpendicular to the connecting rod: this component is also transmitted until the extreme of the crank. There, it can be decomposed in a component pointing towards the center of the motor, which doesn´t create torque, and another one tangential to it, creating torque on the motor.

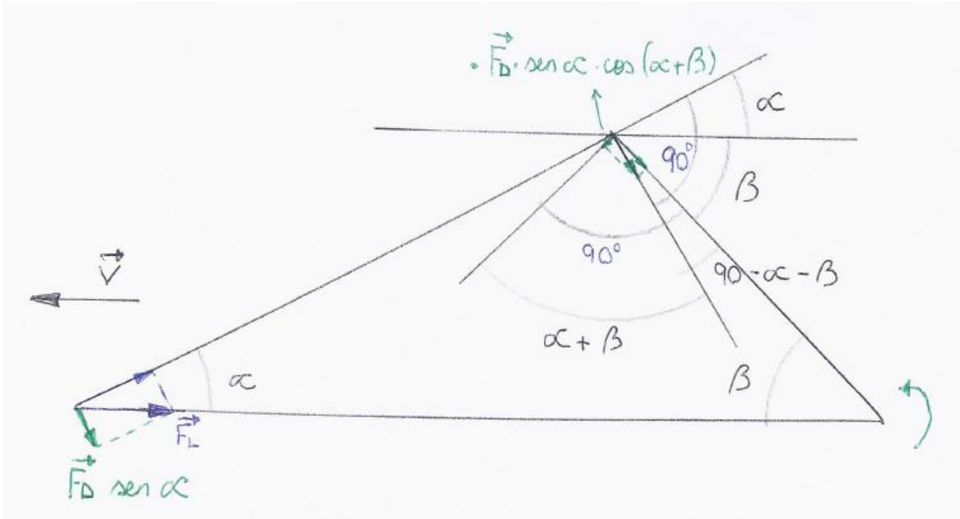


FIGURE 43: EFFECT OF THE DRAG FORCE COMPONENT PERPENDICULAR TO THE CONNECTING ROD ON THE MOTOR TORQUE.

It can be observed in figure 43 that, in this case, the drag force component perpendicular to the connecting rod causes a positive effect on the motor torque, due to the fact that it tries rotating the crank in the same direction that the motor does. However, the significance of this component is much smaller than the other component. This is because the angle α is always lower than 19.47°, and therefore, the drag force component parallel to the connecting rod is always bigger that the perpendicular one.

As this force is acting on the contrary sense to the one defined as positive, this force is expressed as a negative force:

$$F_{Perpendicular} = -F_D * \sin\alpha * \cos(\alpha + \beta)$$

In spite of the fact that, in this draw, this component has a positive effect on the motor, it is not always like this. Imagine that the angle α is 90°. The line perpendicular to the connecting rod in which the final force is drawn would be situated at the right of the crank. Thus, it is easy to watch that in this imaginary case, the final tangential to the motor component would have changed its sense and would be acting against the motor. Therefore, it must have a point where this component provokes a null effect on the motor torque because its value is zero. This point must be the changing point between favorable and unfavorable effects on the motor torque.

This changing point is due to the geometry of the mechanism, so it is easily calculable. In order that the final tangential component doesn't have any influence in the torque, both the line perpendicular to the connecting rod and the crank have to be forced to be coincident. When this happens, it can be perceived in figure 43, that the angle α+β is 90°. Thus:

$$\begin{cases} \dfrac{\sin\beta}{\sin\alpha} = 3 \\ \alpha + \beta = 90° \end{cases} \rightarrow \begin{matrix} \dfrac{\sin\beta}{\sin\alpha} = 3 \\ \alpha = 90° - \beta \end{matrix} \rightarrow \frac{\sin\beta}{\sin(90° - \beta)} = 3 \rightarrow \frac{\sin\beta}{\cos\beta} = 3 \rightarrow \tan\beta = 3 \rightarrow \beta = 71.56°$$

Therefore, the drag force component perpendicular to the connecting rod helps the motor to rotate for motor angles between 71.56˚ and -71.56˚, due to symmetry, and acts against it for the rest of the angles of the circle. Besides, for the angles 71.56˚and -71.56˚, its contribution to the motor torque is 0.

Finally, the total tangential force acting on the end of the crank is the addition of the forces due to the two components. Thus, adding both of the forces I obtain the expression of the total force producing torque on the rotation axis:

$F = F_D * K$ where $K = (\cos\alpha * \sin(\alpha+\beta) - \sin\alpha * \cos(\alpha+\beta))$ is the trigonometric factor

It can exist a doubt about the correlation between the expression and the reality when the effect of the tangential component due to the drag force component perpendicular to the connection rod changes, passing from being in favor to being against the motor. This query disappears if we observe that for β angles bigger than 71.56°, the angles α+β are bigger than 90 ° and as a result, $\cos(\alpha+\beta)$ is negative, changing the sign of this part of the expression.

This demonstration that I have done with the drag force can be also done with the other forces. Nevertheless, this trigonometric factor I have calculated is the same for every force acting on the cylinder movement direction. Accordingly, I am going to apply it also to the lift and the inertia forces.

6.3.6. Fluid dynamic forces' effects on the motor. Total tangential component

It has been demonstrated that there is a trigonometric factor modifying the influence of the forces on the motor. In order to ease the study of the resulting force contributing to the moment of forces, I develop the obtained factor using trigonometric relations. The objective is that the final function only depends on one angle instead of both of them. Thereby:

-1: It is better that the selected angle of dependence is β. That is owing to the fact that the rod whose angle is β, makes circles continually whereas the angle α only takes several values. Thus:

$$\frac{\sin\beta}{\sin\alpha} = 3 \rightarrow \sin\alpha = \frac{1}{3} * \sin\beta$$

-2: These three expressions are going to be used too:

$$\sin(\alpha+\beta) = \sin\alpha * \cos\beta + \cos\alpha * \sin\beta$$

$$\cos(\alpha+\beta) = \cos\alpha * \cos\beta - \sin\alpha * \sin\beta$$

$$\sin\alpha + \cos\alpha = 1$$

-3: Continuing with the trigonometric factor expression obtained before and using the relations 1 and 2:

$$K = (\cos\alpha * \sin(\alpha+\beta) - \sin\alpha * \cos(\alpha+\beta)) =$$
$$= [\cos\alpha * (\sin\alpha * \cos\beta + \cos\alpha * \sin\beta) - \sin\alpha * (\cos\alpha * \cos\beta - \sin\alpha * \sin\beta)] = [\cos\alpha *$$
$$* \sin\alpha * \cos\beta + \cos\alpha^2 * \sin\beta - \cos\alpha * \sin\alpha * \cos\beta + \sin\alpha^2 * \sin\beta] =$$
$$\left[\sqrt{1-\sin\alpha^2} * \sin\alpha * \cos\beta + (1-\sin\alpha^2) * \sin\beta - \sqrt{1-\sin\alpha^2} * \sin\alpha * \cos\beta + \sin\alpha^2 * \sin\beta\right] =$$

$$K = \left(\frac{1}{3} * \sin\beta * \cos\beta * \sqrt{1 - \frac{1}{9} * \sin\beta} + \sin\beta * \left[1 - \frac{1}{9} * \sin\beta^2\right] - \frac{1}{3} * \sin\beta * \cos\beta * \sqrt{1 - \frac{1}{9} * \sin\beta} + \frac{1}{9} * \sin\beta^3 \right) =$$

It can be perceived, that the trigonometric factor consists of two components which can be so-called by the relative position of the forces respecting to the connecting rod, as I have done before:

- Component perpendicular to the connecting rod: Multiplying the forces per this factor, I obtain the effects on the motor of their component perpendicular to the connection rod. I mean, as a result of the previous paragraph, it is proved that this factor transforms the forces acting on the cylinder into a force tangential to the motor, which equivalent to the effect on the motor of the forces components perpendicular to the connecting rod.

$$y = \left(-\frac{1}{3} * \sin\beta * \cos\beta * \sqrt{1 - \frac{1}{9} * \sin\beta} + \frac{1}{9} * \sin\beta^3 \right)$$

Thus, by multiplying it per the fluid dynamic forces, it is possible to observe the behavior of the forces tangential to the motor coming from the lift and drag components perpendicular to the connecting rod, as it can be observe in figure 44:

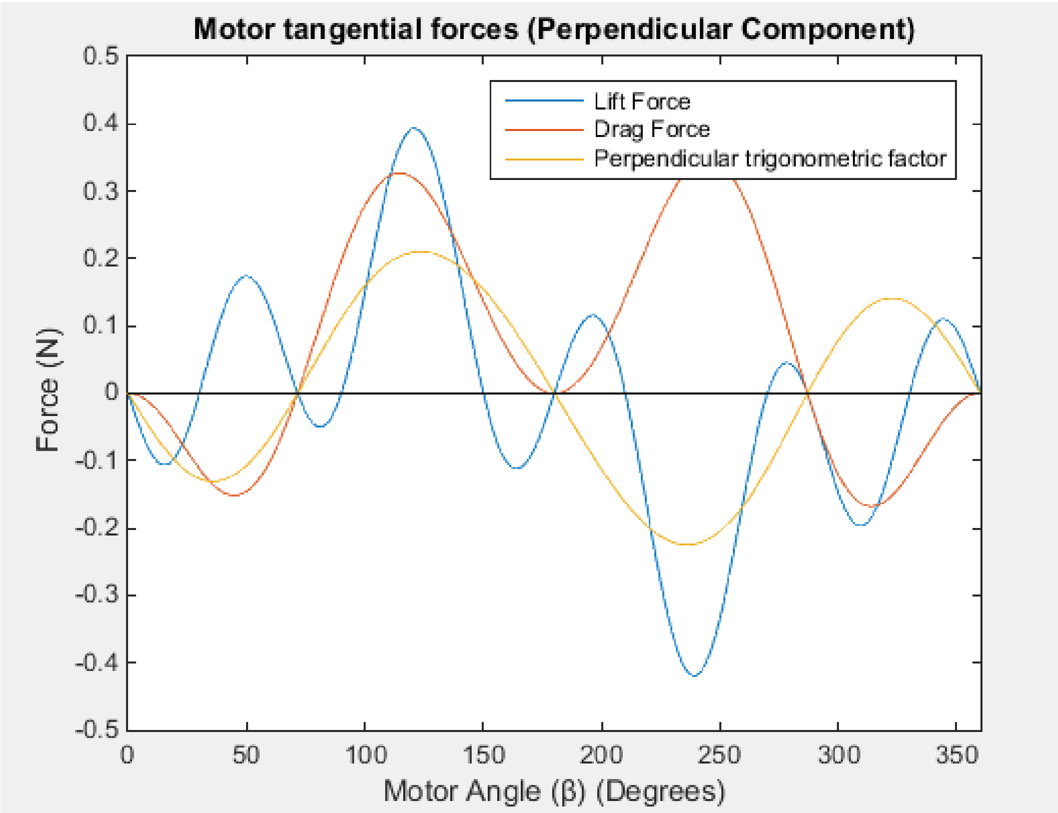


FIGURE 44: IT CAN BE OBSERVED HERE THE BEHAVIOR THE TANGENTIAL PART OF THE TRIGONOMETRIC FACTOR DURING A COMPLETE REVOLUTION.

First of all, it is important to notice that the tangential force to the motor has been drawn opposite to the motor so that the positive values of the forces are against the motor while the negative ones are in favor. As I have said before, it is visible that this component of drag force the force acts in favor of the movement of the motor during this fragment of the circle: β= [71.56˚, 71.56˚]. Nevertheless, during the rest of the time, it acts against the motor motion, ant its maximum value turns up when the angle beta is around 250 degrees. Moreover, it can be also observed that this component is equal to zero when β is 0 or 180 degrees. That can be explained because of the fact that, in these moments, both of the bars of the mechanism are parallel to the drag force and therefore, it is decomposed only in a parallel component whereas the perpendicular one (which gives this tangential force) is zero. Regarding at the lift force, little comments can be said, as its behavior is behavior is different for each water speed. Thus, this behavior of the lift force component perpendicular to the connecting rod is only valid when the water flow is moving at 0.45 m/s, as I assumed before.

- Normal component: Multiplying the forces per this factor, I obtain the effects on the motor of their component parallel to the connection rod.

$$y = \sin\beta * \left(\frac{1}{3} * \sin\beta * \cos\beta * \sqrt{1 - \frac{1}{9} * \sin\beta} + \sin\beta * \left[1 - \frac{1}{9} * \sin\beta^2\right]\right)$$

In figure 45 it is possible to look at the behavior of the parallel trigonometric factor and its impact on the fluid dynamic forces:

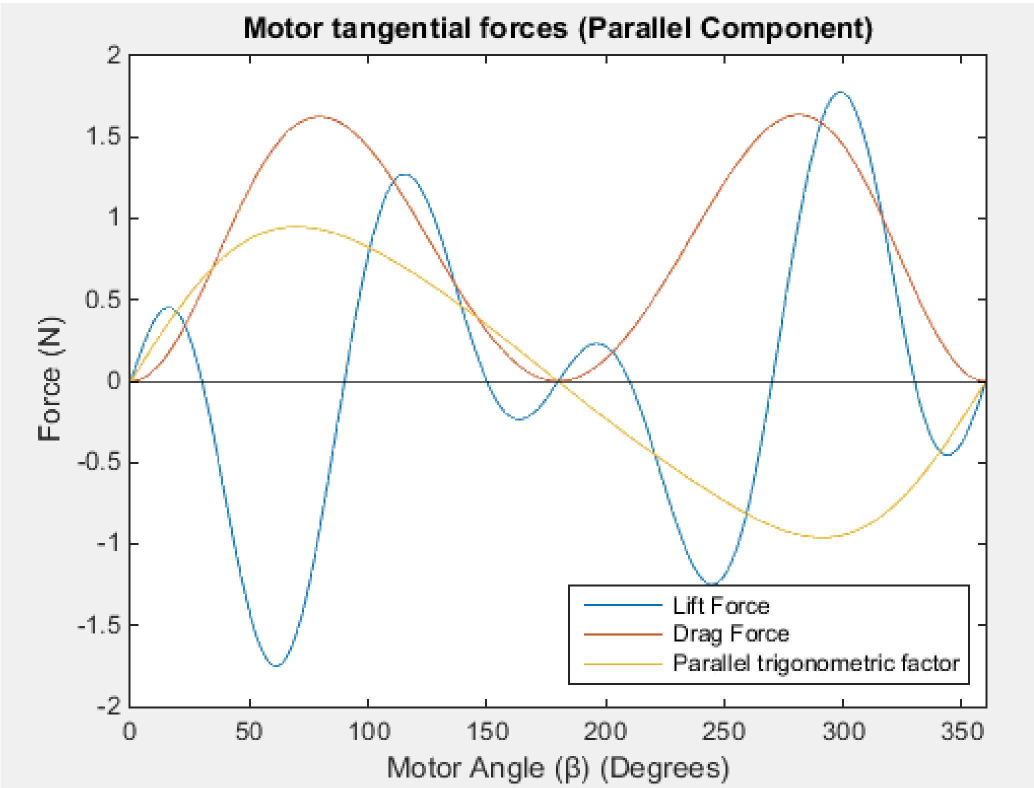


FIGURE 45: HERE, IT CAN BE OBSERVED THE BEHAVIOR THE NORMAL PART OF THE TRIGONOMETRIC FACTOR DURING A COMPLETE REVOLUTION

In figure 45, it can be perceived that this component of the drag force acts always against the motor motion. Its behavior seems symmetric but actually, it is not. Its maximum value appears when the angle beta is around 290 degrees. Furthermore, it can be seen that this component is also equal to zero when β is 0 or 180 degrees. That has a different explanation than before. Although in these moments the drag force is decomposed only in the force parallel to the connecting rod (which causes this component), as the whole mechanism is in line with the motor, this component doesn´t have any influence in the rotational movement of the motor, and so, its trigonometric factor is zero. As in figure 45, the tangential force coming from this lift force component is only valid for the assumed velocity of the water.

However, there are not these separated components but the addition of them which causes the final tangential force on the motor. Thus, I proceed adding up the two components obtaining the equation below:

$$K = \left(\frac{1}{3} * \sin\beta * \cos\beta * \sqrt{1 - \frac{1}{9} * \sin\beta} + \sin\beta * \left[1 - \frac{1}{9} * \sin\beta^2\right] - \frac{1}{3} * \sin\beta * \cos\beta * \sqrt{1 - \frac{1}{9} * \sin\beta + \frac{1}{9} * \sin\beta^3} \right) = \sin\beta$$

There is a curious issue resulting of the addition. I am talking about the fact that the final trigonometric factor doesn´t depends at all on the ratio between the bars lengths. It can be observed that the factor owed to this ratio doesn´t appears on the final equation. It has disappeared during the mathematic simplifications.

In figure 46, it can be observed the total trigonometric factor and its influence in the fluid dynamic forces acting on the cylinder:

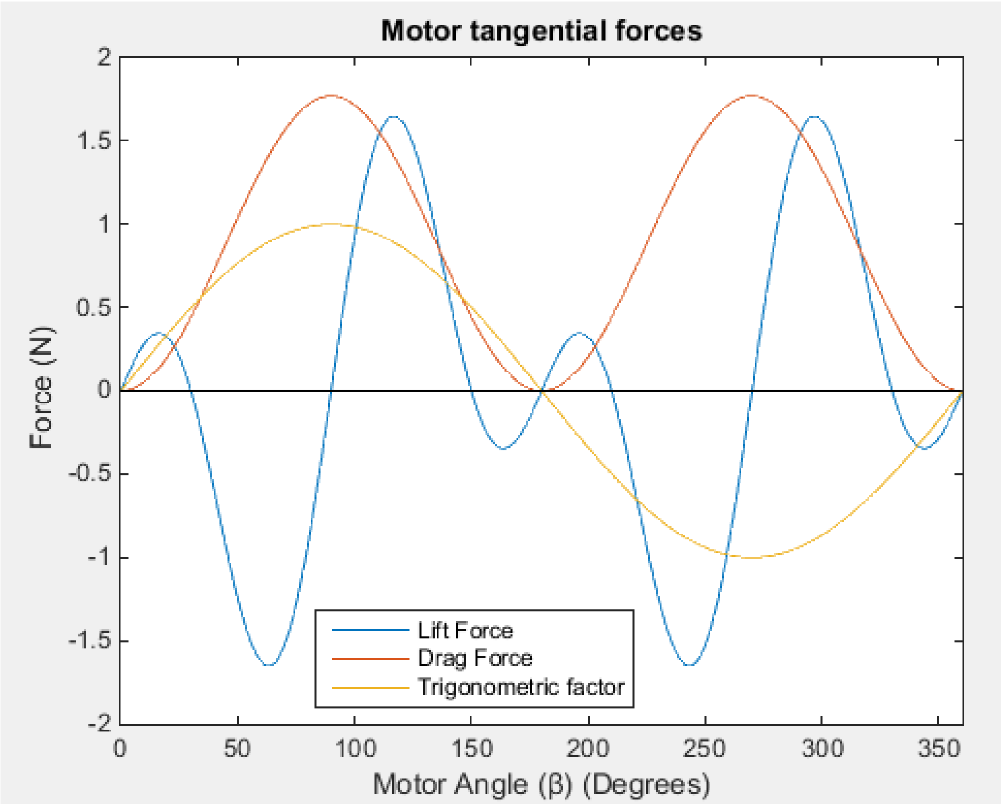


FIGURE 46: IN THIS PICTURE, THE TRIGONOMETRIC FACTOR (YELLOW) AND THE CONTRIBUTION TO THE MOTOR TORQUE OF LIFT AND DRAG FORCES ARE DRAWN.

As it can be observed in figure 46, both in the mathematic equation and in the graphic, the final trigonometric factor affecting the drag force is a sinusoidal function with the same frequency as the motor. Thus, it takes its maximum values when the angle β is 90 and 270 degrees (when the crank is perpendicular to the cylinder path). In these points, the whole drag force is preventing the rotation of the motor whereas, when β values 0 or 180 degrees, there its contribution against the motor is 0.

Finally, I add all the fluid dynamic forces acting on the cylinder to obtain their total impact on the motor torque. It can be seen in figure 47 that the total fluid dynamic force is acting against the motor rotation most of the time:

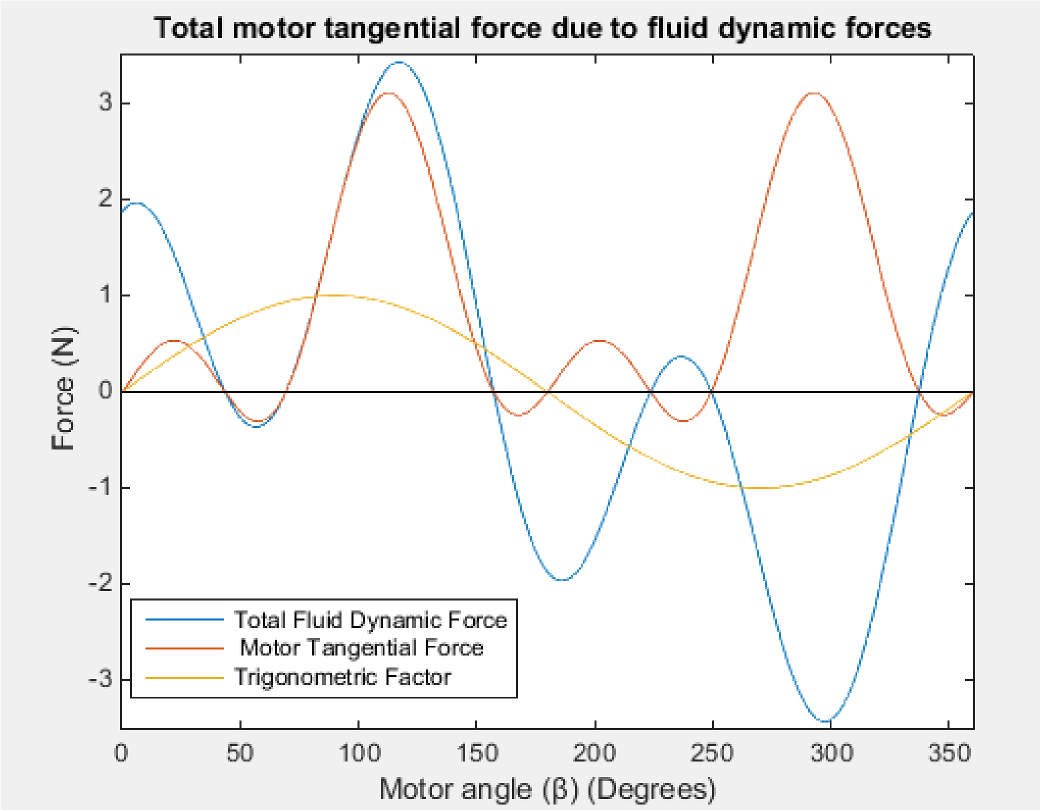


FIGURE 47: TOTAL FORCE ACTING ON THE CYLINDER AND ITS IMPACT ON THE MOTOR.

6.3.7. Lock-In Conditions

During lock-in conditions, the natural shedding frequency falls towards the frequency of the cylinder. As a consequence, beneath this condition and assuming the 2S vortex shedding mode, the lift force has the same frequency of the cylinder. Indeed, its comportment can be completely model: when the cylinder reaches an extreme of the stroke, a vortex is shed and the lift force is maximum, pushing the cylinder to the other side. In this path, the lift force decreases until the center of the stroke, where its value is zero. After the center, the direction of the lift force changes and it starts rising at the same time that it slows down the cylinder. Thus, when the cylinder arrives to the other side, its velocity is zero and the lift force is maximum once more. A vortex is shed and the lift force starts moving the cylinder.

According to this explanation, bellow these assumptions, the lift force can be express as:

$$F_L = 1.8746 \cos\beta \text{ N}$$

The fluid dynamic forces during lock-in condition can be observed in figure48:

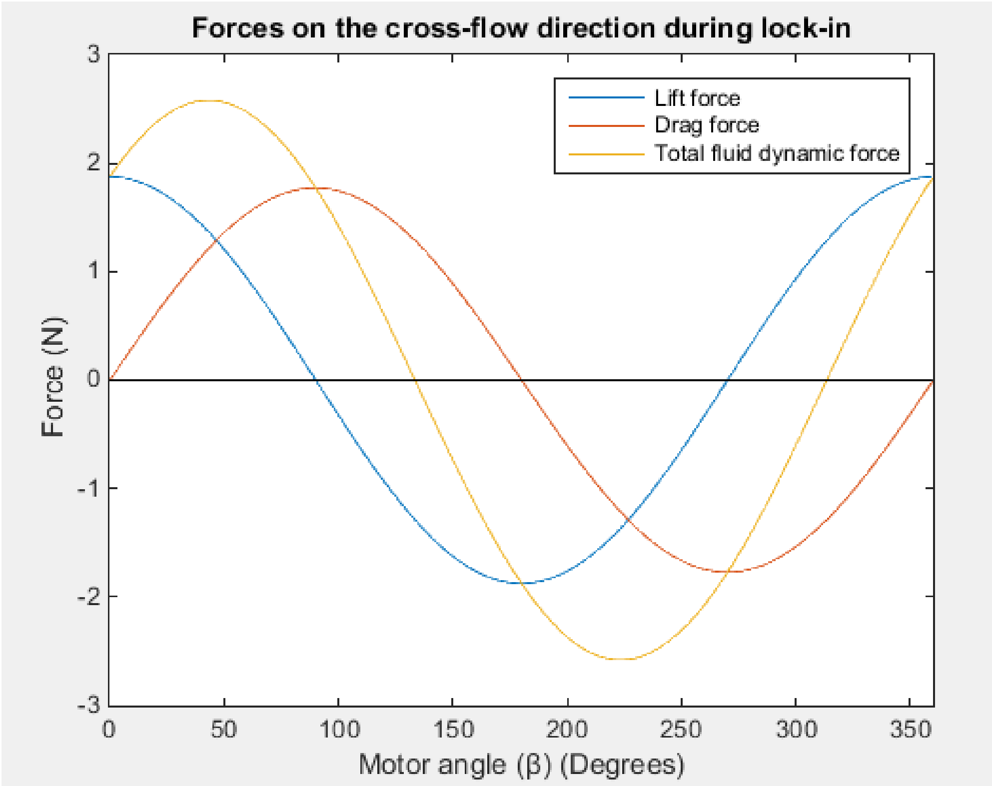


FIGURE 48: FLUID DYNAMIC FORCES ON THE CROSS-FLOW DIRECTION DURING LOCK-IN

It can be observed in figure 49 that the maximum force acting on the cylinder is bigger during non-lock-in conditions than during lock in conditions. Therefore, the total force tangential to the motor coming from the fluid dynamic forces and the required motor torque are bigger during non-lock-in conditions.

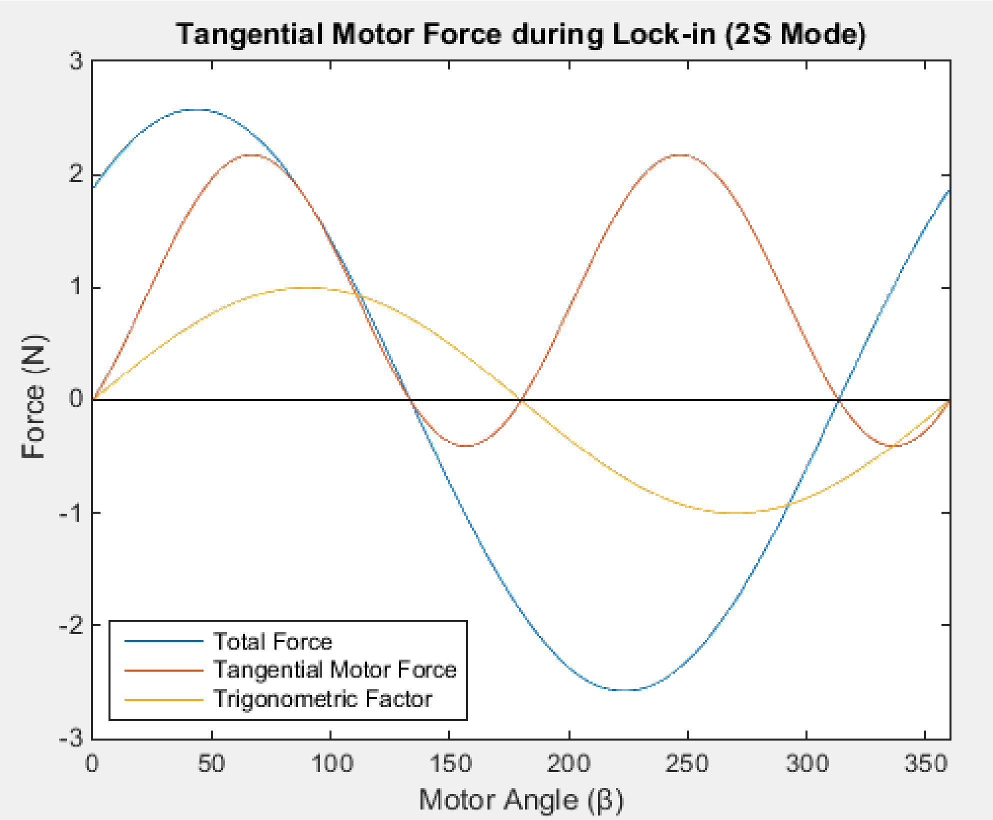


FIGURE 49: TOTAL FORCE ACTING ON THE CYLINDER AND TOTAL TANGENTIAL FORCE ACTING ON THE MOTOR DURING LOCK-IN CONDITIONS.

6.3.8. Inertia Force

The inertia force is the force necessary to accelerate or decelerate a body due to its inertia. Thus, the bigger the mass and the acceleration of a body are the bigger inertia forces are going to appear during its movement (acceleration or deceleration). This force is calculated with the second Newton law:

$$F_I = \sum m * a$$

F_I=Inertia force (N)

m=body´s mass (kg)

a=body´s acceleration (m/s^2)

In this case, the moving body is the cylinder and the box surrounding it and its mass is around 0.45 kg. Hence, knowing that the acceleration of the cylinder is $a = -1.973 * \cos\beta$, the inertia force is:

$$F = -\sum m * a = -0.45 * \ -1.973 * \cos\beta \ = \ 0.8878 * \cos\beta \ \ \mathrm{N}$$

It can be observe in figure 50 that the inertia force is sometimes helping to the motor and sometimes obstructing it. Indeed, it can be noticed that the inertia force is acting against the motor when the cylinder is leaving from the ends of the stroke and in favor when the cylinder is arriving to them. It also can be observed that the effects of the inertia force on the motor are reduced by the trigonometric factor. This is because when the inertia force takes its maximum values on the ends of the stroke and it is zero on the center whereas the trigonometric factor is zero in the extremes and maximum in the center of the stroke.

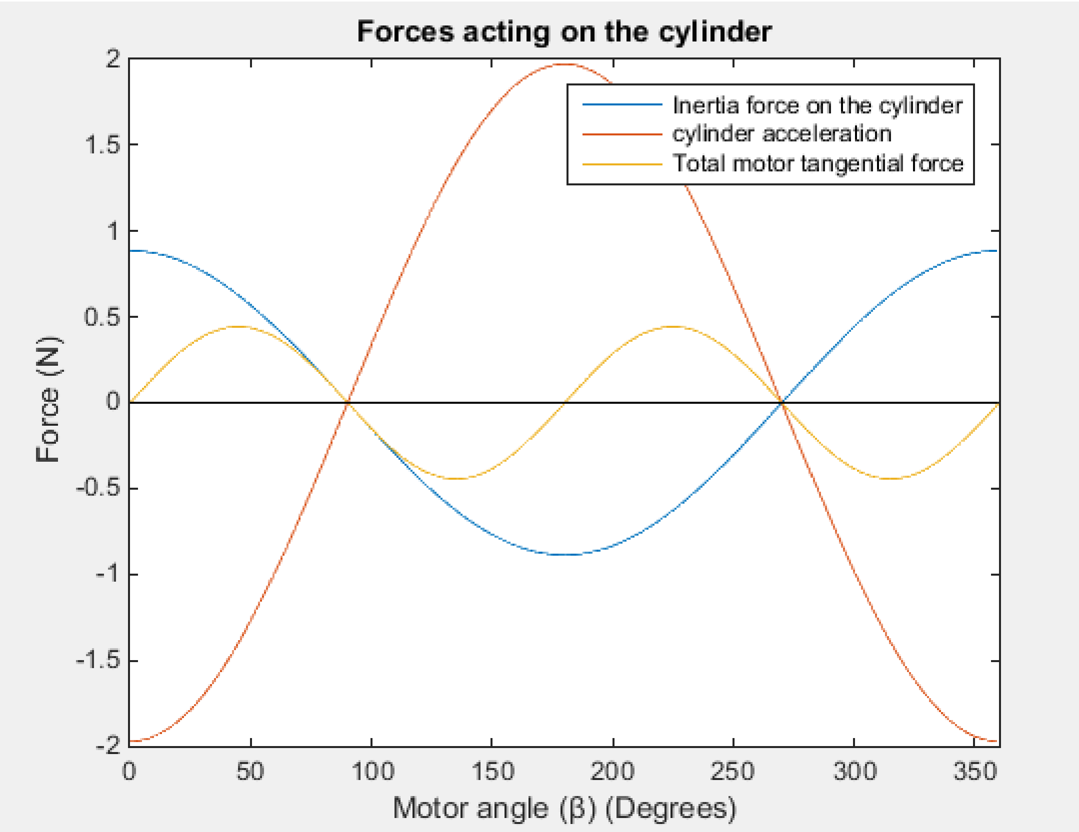


FIGURE 50: INERTIA FORCE AND THE FORCE TANGENTIAL TO THE MOTOR CAUSED BY INERTIA.

6.3.9. Motor Inertia

It needs to be said that there is another inertia force acting on motor, which is the inertia due to the motor angular acceleration. As the mechanism has been designed to have a constant angular velocity, it only affects to the motor during the transitory state. It can be calculated with a variant of the Newton second law:

$$\tau = I * \alpha$$

τ=Motor torque (N*m)

I= Inertia of the load on the axis of the motor ($\mathrm{kg*m^2}$)

α=Angular velocity of the motor (rad/s^2)

In order to calculate the inertia of the moving load, I am going to suppose that it only consist on a cylinder. The moving of inertia of a cylinder from its center is:

$$I = \frac{1}{2} * m * R^2$$

However, I want the inertia with respect to the center of rotation of the motor. Thus, I applying the Steiner equation I obtain the following equation (where L= distance from the cylinder to the motor center of rotation):

$$I = \frac{1}{2} * m * R^2 + m * L^2$$

Knowing that m=0.45 kg and that the maximum length is the addition of the measures of the crank (0.06m) and the connecting rod (0.15m), I can calculate the maximum inertia of the cylinder with respect to the motor centre of rotation:

$$I = \frac{1}{2} * 0.45 * 0.015^2 + 0.45 * 0.21^2 = 0.02 \text{ kg} * \text{m}^2$$

Knowing that the maximum frequency of the cylinder is 3Hz and supposing that it is wanted to be achieved in two second, I can calculate the angular velocity and the maximum torque of the motor:

$$\omega = 3Hz * \frac{2\pi}{1Hz} = 6\pi \ \rightarrow \alpha = \frac{\omega}{t} = \frac{6\pi}{2} = 3\pi \text{ rad/s}^2$$

$$\tau = I * \alpha = 0.02 * 3\pi = 0.1884 \text{ N} * \text{m}$$

This is the maximum torque of the motor due to this inertia, and it only affects to motor during the transitory state.

6.3.10. Motor Torque

In order to calculate the motor torque needed during the transitory state, I use the maximum moment of forces produced by the forces acting on the cylinder, because even if in the majority of the time the lift force opposes less resistance, the motor has to be prepared for surpassing every torque at every time. As I have said, this maximum torque takes place during non-lock in conditions and is calculated reckoning the velocity and the radius of the crank as the largest values that is possible (v=0.45 m/s and r= 0.06 m).
The tangential force acting on the motor coming from the fluid dynamic force is defined as force per meter of immersed cylinder. Considering that the cylinder is immerged 0.15 meters, I add their effect to the effect of the inertia force:

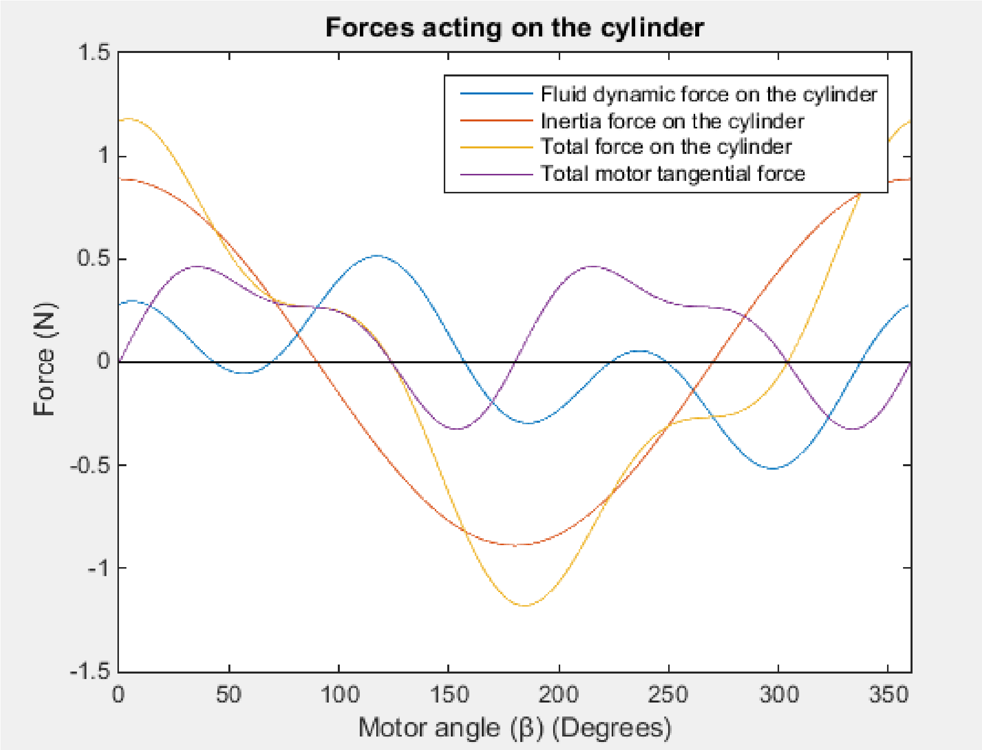


FIGURE 51: TOTAL FORCE TANGENTIAL TO THE MOTOR.

Looking at figure 38 it can be observed that the biggest tangential force acting on the motor is 0.85 N. Knowing that and taking as radius, the maximum value of the crank (0.06m), I can calculate the maximum motor torque applying a security factor of 1.5 and using the following equation:

$$\tau = F \; x \; r$$

F= Tangential force (N)
r= Distance from the force to the center of rotation (m)

T= motor torque (N*m)

$$\tau = 1.5 * F \times r = 1.5 * 0.48 * 0.06 = 0.043 \; N * m$$

Once the motor torque is known, it is possible to calculate the required motor power a frequency of 1 Hz ($2\pi \frac{rad}{s}$):

$$P = \tau * \omega = 0.043 * 2\pi = 0.27 \; W$$

ω= Angular velocity of the motor (rad/s)

As a result, the required motor power is really low, and the effect of the forces acting on the cylinder is almost negligible. However, there are three reasons why I take a bigger motor:

- Firstly, I have done the calculation for a cylinder frequency of 1 Hz and actually it can be until 3Hz. Multiplying the angular velocity per three increases until nine times the value of the inertia force and the drag force, as inertia depends on the acceleration, which depends on the square angular velocity of the motor, and the drag depends on the square cylinder velocity, which depends directly on the angular motor velocity. Thus, it can be estimate a torque nine times bigger than the torque for 1Hz, and accordingly, a power twenty seven times bigger.

- Secondly, in the transitory state and extra torque is required for the acceleration of the motor (maximum extra torque required: $\tau = 0.1884\ N * m$)
- Finally, I have neglected the effects of the friction on the mechanism, but it is possible than in the reality, the bearings don´t have the smooth behavior they are supposed to have and the power required by the friction force is not negligible.

That is why, I finally select the stepper motor SY57STH51-4004A which has a torque around 0.6 N*m for the range of cylinder velocities of this project.

7. Automatic control

Thank to the driving mechanism, the cylinder is able to move upwards and backwards. Yet, not any kind of movement is good enough for this experiment. As I have said before, the cylinder has to move following a determinate function and at determinate speeds. The first one is fixed by the driving mechanism whereas the second one has to be controlled by automatic control.

An automatic control in nothing more than an electronic device connected to an actuator. The performance process of a simple automatic control is explained straightaway: any person using an electric motor introduces the value of the characteristic they want to supervise in the electronic device. It, thank to its electronics, transforms this value in an instruction for the actuator, which is the device, or element of the control, that makes physical stimuli for changing the controlled characteristic. This is the classic automatic control, called open-loop control, due to the fact that there is no way of knowing if the plant is doing what you wanted or not. As a result, this kind of control is almost always accompanied with a lack of precision. In order to enhance it, another kind of automatic control appeared afterwards: close-loop control, which is based on a continuous flux of information between the control and the plant.

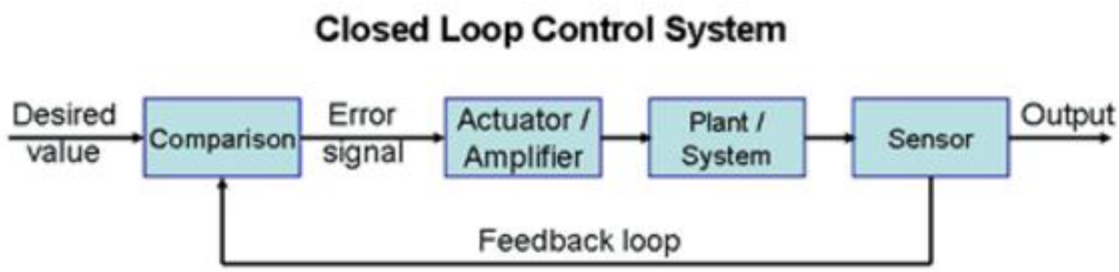


FIGURE 52: CLOSE-LOOP CONTROL SYSTEM.

According to figure 52, every automatic control system consists of these elements:

- Plant/System: The plant is the physical system whose characteristics I want to control. In this case, we have two plants: I want to control the velocity of the water flow and the velocity of the cylinder.

- Sensor: As I have said, a close-loop control has much more precision that an open-loop one because it actuates regarding to the information obtained from the final characteristics.

Thus, as an indispensable requirement, this kind of control has to have an appliance for measuring the controlled characteristic at the end of the process. This device is commonly called sensor and it is only present in close-loop automatic control systems.

- Control (Comparison): The control of the circuit is the set of operations that reckon the signal that has to be sent to the actuator. It can be thought as the intelligent part of the system, as it calculates the stimulus necessary to get a result from a known value coming from direct people orders (open-loop control), or from a sensor (close-loop control) . As it can observed in figure 52, in close-loop controls, that the sensor measures and sends information concerning the controlled characteristic to an implemented electronic circuit (control), where it is compared with its desired value. From this comparison, the error value is obtained. This error is the difference between the current value that there is, and the one desired. Due to this difference, the control calculates the analogical signal that it has to send to the actuator in order that it changes its performance to a new one that allows accomplishing the desired value of the controlled characteristic, always trying making the error equal to zero.

  There are two common ways of setting up this part of the automatic control. Usually, if it is a frequently used automatic control system, its electronics are sold already implemented in unique piece. Otherwise, there exit software and implemented chips that allow making control circuits handily. In this project, as it is a small particular application, I am going to implement it using the opened software Arduino Uno. This software permits making almost any sort of controlling program by building and external circuit with electronic components and writing a set of commands in an informatics program.

- Actuator: Every automatic control has an actuator, which is the piece in charge of doing whatsoever which is necessary to change the characteristic that is being tried to control. Thus, while the control is though as the intelligent part of the system, the actuator must be seen as the practical changer of the controller characteristic. It is true that every calculation have been already done in the previous step, but it is thanks to the actuator that the plants characteristics change, because it translate electric signals into physical stimuli for the systems.

Regarding at this particular project, it has to be notice that, on this occasion, having a steady velocity is very important because a high precision is required in VIV rehearsals. Therefore, controlling the motor one with an open-loop control is not accurate enough, because this kind of control does not have any kind of feedback. As a matter of fact, using it is easy and quite risky, because it is impossible to be aware of whether the output velocity corresponds accurately with the one introduced in the control or not. Thus, in order not to jeopardize the results of the experiments, I would utilize a close-loop control in both of the motors that are going to be used. On the contrary to the open-loop control, this control is based on feedback. It takes information concerning the final motor velocity and compares it with the wished values, which have been introduced in the electronic device before. Depending on the difference between the taped value and the feedback, the device sends different commands to the actuator, which makes the motor change its movement.

I have said that I would utilize it because, actually, I use an open-loop control. The main reason why I do it is because I don't have enough time to do it. This deepening in the automatic control is outside of my project limits. Nevertheless, I have fathomed a little bit in the subject and I am going to mention it and give a piece of advice to future people working on the subject, but, anyway, it has to be clear that I have not utilized the sensor and the feedback that I am going to talk about.

As far as the design is concerned, it has two different motions: the water motion and the cylinder one. Obviously, each of them has to be moved with different motors, and, therefore, two different automatic controls are needed; one for controlling the velocity of the water flow and the other one for controlling the cylinder velocity.

7.1. Water flow Control

The first feature which I have to control is the velocity of the water flow. In this case, the actuator is going to be the mechanism that relates the brushless motors to the propellers. More specifically, the Arduino (control) sends some signals to the motors that make them move in the way desired, and this movement is transmitted thanks to the pulley mechanisms until the boat propellers, which push the water downstream.

Actually, the Arduino does not send signals directly to the motors, as they are not capable of reading anything. Indeed, the control of the motors is done thank to small electronic converter whose principal functions are receiving the instructions of the Arduino and transforming the incoming energy from the power supply so that the motor moves as the Arduino´s signal commands. It is a little tricky to understand. Therefore, I am going to give a little piece of information about brushless motors, before explaining it.

7.1.1. Brushless Motors

A brushless motor is a synchronous motor which has permanent magnets in its rotor instead of electric coils, so that brushes are eliminated, as its own name said. As every motor, its rotation is due to the presence of two magnetic fields, one caused by the stator coils and the other one by the permanent magnets, which are usually made in NE-B-Fe. Both magnetic fields cause electromotive forces on the other component and there is a torque resulting of the vector product of these two forces that tries putting these fields together. As a result, the rotor spins to line up both magnetic fields. Therefore, the key of this kind of motor performance is supplying the stator with three sinusoidal currents outdated 120 electric degrees so that it generates a spinning magnetic field that rotates at the same velocity of the rotor.

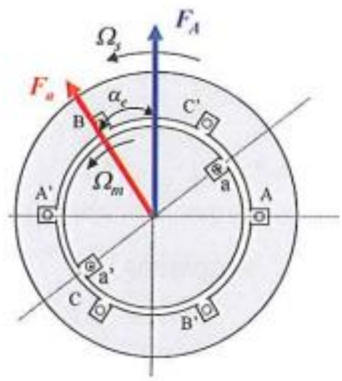


FIGURE 53: ATTRACTION BETWEEN TWO MAGNETIC FIELDS, DUE TO THE ELECTROMOTIVE INDUCED FORCES. IT CAN BE OBSERVED THAT THE ROTOR ROTATES TOWARDS THE STATOR MAGNETIC FIELD, TRYING TO LINE UP BOTH MAGNETIC FIELDS. THE SAME HAPPENS WHEN THE ROTOR IS MADE WITH PERMANENT MAGNETS.

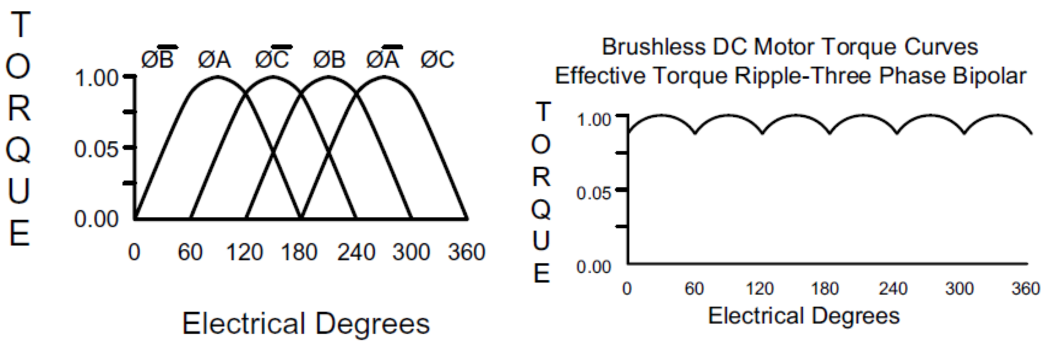


FIGURE 54: TORQUE PRODUCED BY EACH PHASE OF THE STATOR (LEFT) AND EFFECTIVE MOTOR TORQUE (RIGHT)

Thus, as long as this magnetic field rotates at the same velocity than the rotor, there will be a difference between the angles of both fields and the motor will continue spinning. As a consequence, it can be observed, that the velocity of brushless motors is fixed by the frequency of the incoming electric waves supplying the stator. In addition, it also interesting that, as the torque is a result from a vector product, all motors though to have a 90º angle between the stator and the rotor magnetic fields.

7.1.2. Electronic Converter (Frequency driver)

As I have said before, every automatic control has an actuator, which is the mechanism in charge of doing whatsoever which is necessary to change the characteristic that is being tried to control. In this case, the actuator needs a frequency driver. This frequency driver permits controlling the frequency of the motor rotation, and, consequently, the velocity of the cylinder. It does it receiving evaluating and transforming the information coming from the Arduino. In addition, it allows connecting the motor with the power supply, without touching the Arduino, which only withstands 5 volts and 40mA, much less than the motor requirements.

A frequency driver is an electronic device whose performance is based in the rectification of the electric current and its posterior modulation. As it is possible to observe in figure 55, this device consists of a rectifier circuit, a DC link, and a inverter circuit.

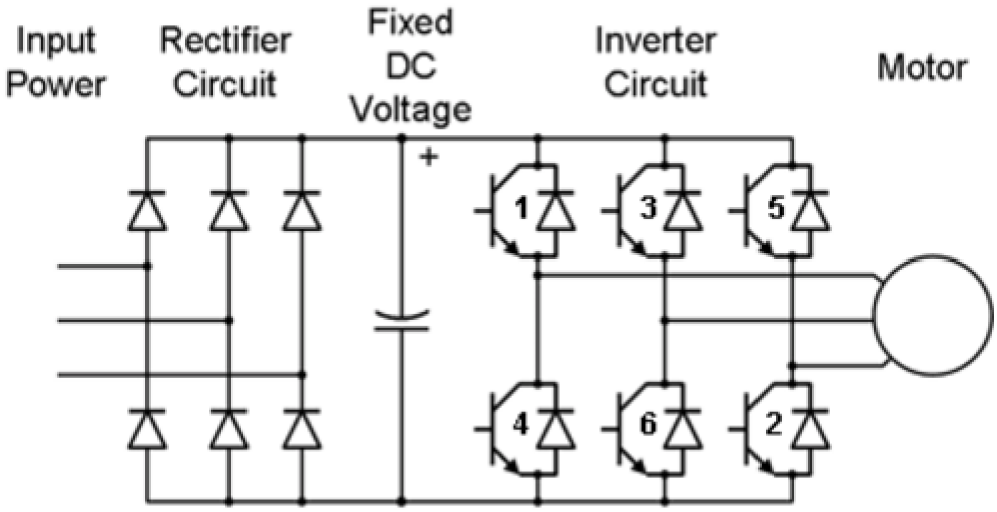


FIGURE 55: FREQUENCY DRIVER.

The operation of this device can be explained separately with the performance of all of its components:

- **Rectifier Circuit**: The rectifier circuit is only implemented with a diodes bridge and its mission is nothing but transform the AC current coming from the plug into a DC current.
- **DC Link**: The most important component of the DC link is a capacitor, whose mission is stocking energy by acting as a depositor. In order that the frequency driver acts correctly, the DC link has to be between a minimum and a maximum voltage:
    - **Minimum voltage**: If the DC link is under this minimum voltage, the frequency driver cannot supply the energy necessary to the motor.
    - **Maximum voltage**: If the DC link is up to this maximum, it could be thought that a DC link is giving too energy to the motor due to an overload, but it is hardly ever like this, because the device has different components and resistances to prevent it. Accordingly, this overvoltage is usually caused by the fact that the motor is no longer acting as a motor but as a generator.
- **Inverter Circuit**: This part of the circuit is composed of 6 transistors and 6 freewheel diodes, and it is aimed to modulate the electric current as you like. In order to do this, it utilizes Pulse Width Modulation (PWM), which is a technique that allows obtaining almost any kind of electric wave by using micro pulses.
  As it can be observed on the figure 56, PWM performance consist of combining thousands of micro pulses whose average generates an electric wave behaving as we wanted. Micro pulses are repeated once every period, which is between hundreds or thousands of Hertz, and they can only take two values: high (DC voltage) or zero. However, the voltage of the final wave is usually a sinusoidal wave, taking every voltage value. This is possible because it not depends on each pulse but on the average of their duty cycle. The duty cycle refers to the percentage of the period time that the pulse takes the high voltage value. Accordingly, the local value of the output function corresponds to the average of the duty cycles of the pulses in this local interval. Regarding at the frequency of the output wave, it depends how quickly the duty cycle of the pulses varies with the time.

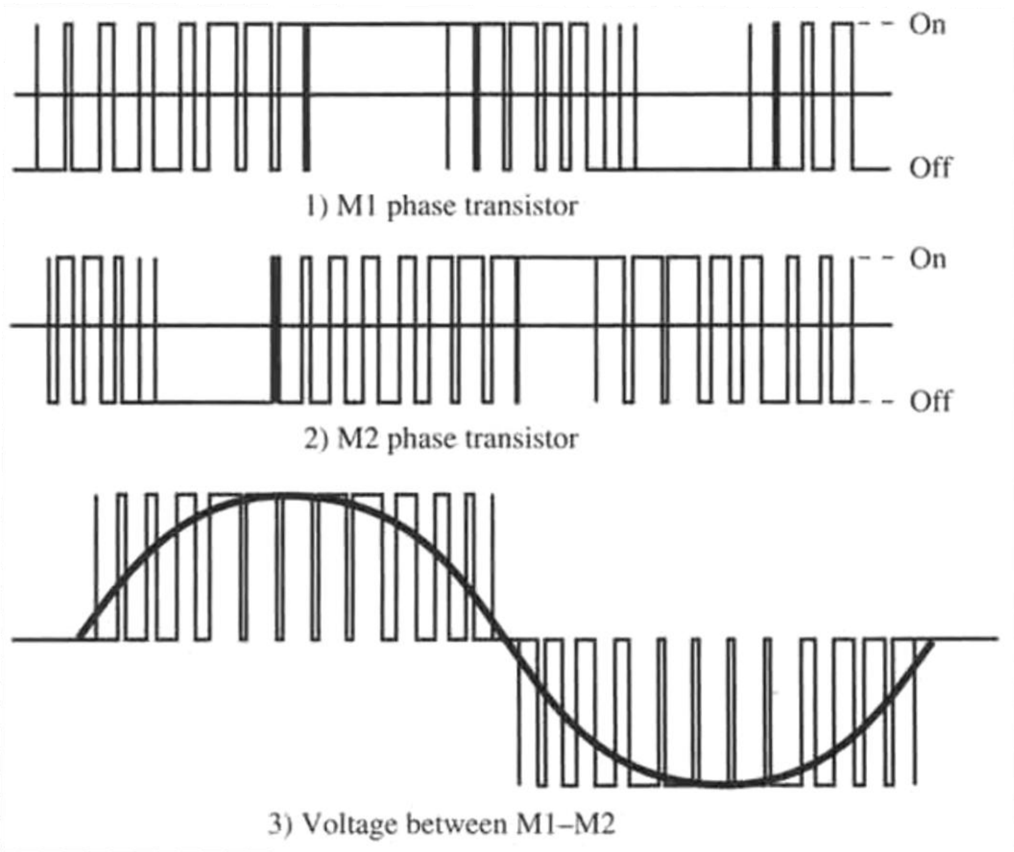


FIGURE 56: PULSE WIDTH MODULATION

All these setting is possible thanks to the six transistors place on the inverter circuit. Each two of them supply one electrical phase of the motor. Why are necessary two of them and not only one? That is because the transistors only are able to conduct the current in one direction. Thus, as the output sinusoidal way has to have positive and negative voltage values, there have to be two transistor acting together to form it. One of them gives the positive voltages whereas the other one gives the negative ones, as it can be seen in the 42nd picture. The function of the diodes consists of protecting the transistors from residual currents coming from the motor, as it is an inductive load.

Particularly, in this design, two brushless motors are supplied by a small electronic converter (Afro Esc). Therefore, as there are four motors, I need two electronic converters. These electronic converters are a little bit different than the theoretical one, as they are directly supplied by a battery (DC voltage). Hence, they don´t need the rectifier circuit.

7.1.3. Control: Arduino commands

Finally, in order to control the velocity of the water, I only need to make a little circuit connecting the power supply, the electronic converter and the motors, and to write a set of commands for controlling the electronic converter.

In this case, the connecting circuit is going to be easy, as the electronic converter has two wires which are thought to be connected automatically with the power supply, and three more to be connected with the three stator phases of the motor. In addition, it has another little set of three slim wires, which are thought to be connected with Arduino (voltage, ground, and information respectively) as it can be observed in figure 57.

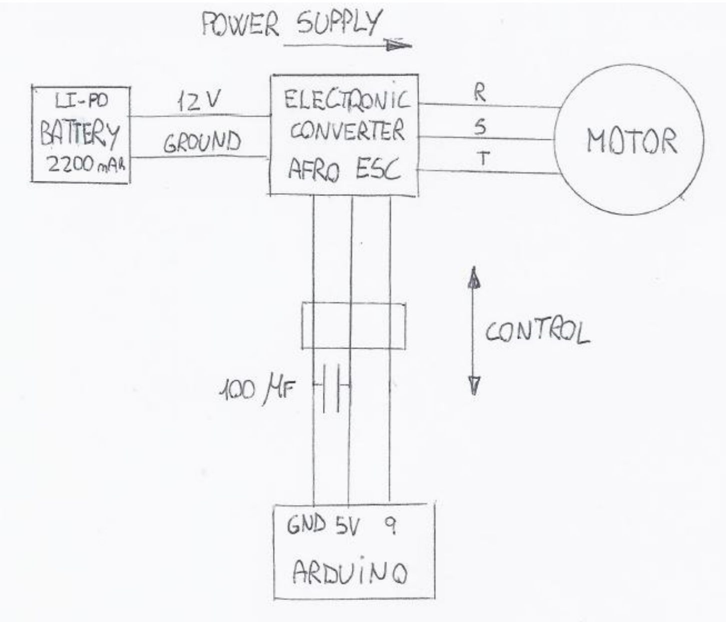


FIGURE 57: BRUSHLESS MOTOR ELECTRONIC CONNECTION.

As I have said before, brushless motors velocity only depends on the frequency of the electrics waves which supply the stator coils. Nonetheless, this characteristic of the brushless motors is usually given by a feature called kV, which makes reference to the number of revolution per minute which the motor rotates per unit of voltage applied when it is acting without load. Thus, it seems that the velocity of the motor only depends on the voltage which it is supplied with. However, it is not true. Actually, the velocity of these motors are expressed in this way because they are always supplied with an electronic converter which always maintains constant the quotient between the voltage and the frequency of the electric waves that it produces. This is due to the fact that the motor torque is directly proportional to this squared factor, and therefore, to vary the velocity while keeping constant the motor torque, it is necessary maintaining the relation between these two magnitudes.

In order to change the motor voltage and the frequency at the same time, it is necessary programming a set of commands in Arduino that tells the converter the voltage it has to send to the motor. In order to do it, it is necessary to take into account that the converter behaves like a servo motor. That means that the controller can be interpreted as a gate, that can be closed, not letting passing any voltage (0 degrees), or opened, allowing supplying the motor with the highest voltage (180 degrees).

The way of controlling it is sending to the controller an analogical pulse-width-modulated signal from Arduino. It needs to be pointed that the pulses composing this signal are not continually changing as the ones of the electronic controller, but they are always equal, maintaining the duty cycle until a new order is given to Arduino. Therefore, the higher the duty cycle of these pulses is, the higher the amplitude and the frequency of the sinusoidal voltage waves exiting the converter is. The intensity coming through the power supplied is determined by the load the motor is moving. Thus, the bigger the load is, the bigger current intensity it requires from the power supply.

Thus, I make an informatics program, which is attached at the end of the document. This program allows the experimenters entering the desired vortex shedding frequency by writing it in the keyboard. Once the frequency has been introduced to the program, Arduino acts sending a pulse with modulation wave to the controller, which behaves like a servomotor. Thus, in my Arduino program, I set a range of angles of the servo (150˚-180˚) related with a range of frequencies (0-3 Hz), so that each frequency value is related with an amplitude of the voltage wave sent to the motors, and therefore, with a velocity of the propellers. For instance, using my program, if the selected frequency is 2 Hz, the servo motor will drive an angle of 170˚, so the amplitude of the motors voltage will be 170/180=0.94 times the nominal amplitude, and hence, the velocity of the motor will be 0.94 times its velocity for the same load. In addition, the program allow stopping the motors whenever you want by only pressing ‘x’ on the keyboard.

It need to be said that this program is not exact at all, because this relation between this ranged is a rude approximation. However, the correct relation only can be known experimentally, doing some test to obtain it. This is because this relation depends on how the motors quickly the motors spin below load and on how this load varies with the velocity of the water.

It can be observed that I relate the velocity of the motors and the vortex shedding frequency utilizing an arbitrary factor. This factor is not accurate enough to make them rotate at the correct velocity. Indeed, is may be wrong. Yet, it is enough as a first approximation to the correct relation between them, which must be determined by doing some test of the water flow behavior after building the circuit.

It also has to be said that a brushless motor should have a rotation velocity below load between the 70% and the 80% of its velocity feature Kv. Therefore, if they spin more slowly, it means that the propellers, and accordingly, the loads, are too big for this motor. If this happens, it would be necessary changing the design, and supplying each propeller with a different motor instead of utilizing only one motor to supply two propellers, as I am doing now.

7.2. Cylinder Velocity Control

The second feature which needed to be controlled in this project is the velocity of the cylinder. The driving mechanism I have designed acts as actuator in this control, as it transmits the changes in the motor rotation velocity to the cylinder, changing its velocity too.

In the previous automatic control I have use Arduino and an electronic converter to control the motor. Likewise, I utilize Arduino once more to control the cylinder velocity. However, I change the brushless motors which I have used before for a stepper motor. Accordingly, I don´t use an electronic controller any more, but a stepper motor driver.

7.2.1. Stepper Motor

As I have calculated before, the cylinder oscillations frequency has to be between 1 and 3 Hertz. As the cylinder is connected with the motor by a simple crank-connecting rod mechanism, one oscillation of the cylinder corresponds to one motor revolution. Accordingly, I want the motor to spin with a velocity between the following limits:

$$1Hz * 60 = 60\ rpm$$

$$3Hz * 60 = 180\ rpm$$

Thus, the velocities of rotation are very low. In addition, I want the movement to be quite precise. Therefore, as stepper motors ensure accuracy in little mechanism motions and provide big torques spinning at low velocities, why I select a stepper motor for this control.

A stepper motor is an electro mechanic device that transforms electric impulses on discrete angular displacements. This is due to its internal structure. This kind of motor usually consists of a stator with different numbers of coils and a rotor with permanent magnets. Its performance is based on the controlled electric supply of its coils. When one of its coils is supplied, it generates a magnetic field that interacts with the magnetic field of the magnets causing a several-degree rotor rotation. This discrete number of degrees is called step, and it depends on the number of steps per revolution of the motor, which is one of its principal characteristics. Carrying on with the explanation, this rotor one-step rotation causes the lining of both magnetic fields. Therefore, in order to continue the rotor movement, it is necessary to supply the next coil, and once the motor rotates, the next one, and then supplying successively each coil of the stator, so that the sum up of all the discrete motor rotations becomes only one continue movement. This supply cannot come directly from the electric net or a battery, but it has to be correctly controlled by an external device, concretely a stepper motor driver.

Particularly, in this project, I use a SY57STH51-4004A stepper motor. As it can be seen in the characteristics brochure (attached at the end of the document), this motor holds a torque of 140 N*cm, bigger than the one I need. However, is has to be said that this kind of motor is featured for the decreasing of its torque when its velocity rises. Therefore, it would be necessary calculating the velocity (steps per second) required by the mechanism and looking at the torque provided by the motor in this condition. Thus, the highest velocity I need is:

$$180\ rpm\ \rightarrow 3Hz\ \rightarrow 3\ revoltions\ per\ second * 200\ steps\ per\ revolution = 600\ pps$$

It can be observed in the motor torque graphic in the motor features brochure that the motor torque at this velocity is around 0.6 N*m, which is enough for reaching every working point of the design.

7.2.2. Control: Arduino commands

To control the velocity of the stepper motor, I use a stepper driver, because it makes the control of the motor easier and it is able to provide the nominal intensity required for the motor, which is much higher than the one that Arduino can withstand. Nowadays, stepper drivers are implemented in a really comfortable way to the user, because they only have to connect every wire in place and send from Arduino a digital signal changing continually from 5V to 0V.

Therefore, the only issue I have to care for is the connections of the wires, which can be observed in figure 58. This driver (HY-DIV268N-5A) is prepared to be connected to a bipolar stepper motor, which is a kind of motor which only have two different sets of coils. In order to make the connections, this driver has twelve pins, whose function is going to explain bellow:

- **Direction pins:** There are two direction pins. They are prepared for changing the sense of the motor rotation. One of them has to be connected to ground and the other one to one Arduino digital output. Depending if this output sends high or low voltage values, the motor rotates clockwise or counterclockwise. As in this project, both spinning senses has the same effect on the cylinder movement, I don’t really care about these pins.
- **Signal pins**: They are the pins that really fixe the motor rotation. There two pins, one of them connected to ground and the other one to one Arduino digital output. While the previous pin values can be set at the beginning of the program and not be touched any more, this signal pin has to send periodic signals to the driver, changing its value form high voltage to low voltage and vice versa every time. This is because this pin is the one that truly supplies the coils. Therefore, these changes are aimed to switch on and switch off the different coils. It is the driver which is programmed to decide which coil has to be supplied each time. In addition, this signal is responsible of the control of the motor velocity, which can be controlled varying the pin twinkle timing. Indeed, the quicker it changes, the quicker the steps take place and the quicker the motor spins.
- **Enable pins:** Once more, there are two pins; one of them connected to ground and the other one to a digital output from Arduino. This could be regarded as a security pin, owing to the fact that the motor only rotates if this pin is activated.
- **Coils pins:** As this driver is prepared for bipolar stepper motors, it has four pins to connect to each of the extremes of both of the two stator sets of coils. It needs to be remarked that this connections has to be correctly done, because if they are wrong, the motor would not rotate, but it only vibrate. If this happens, it is because the coils are badly connected and therefore, the driver is supplying to the erroneous coils.
- **Power supply pins:** These two pins have to be connected with the voltage and the ground of the external power supply. They are absolutely necessary because Arduino cannot provide the current and the voltage necessary to obtain the nominal motor torque.

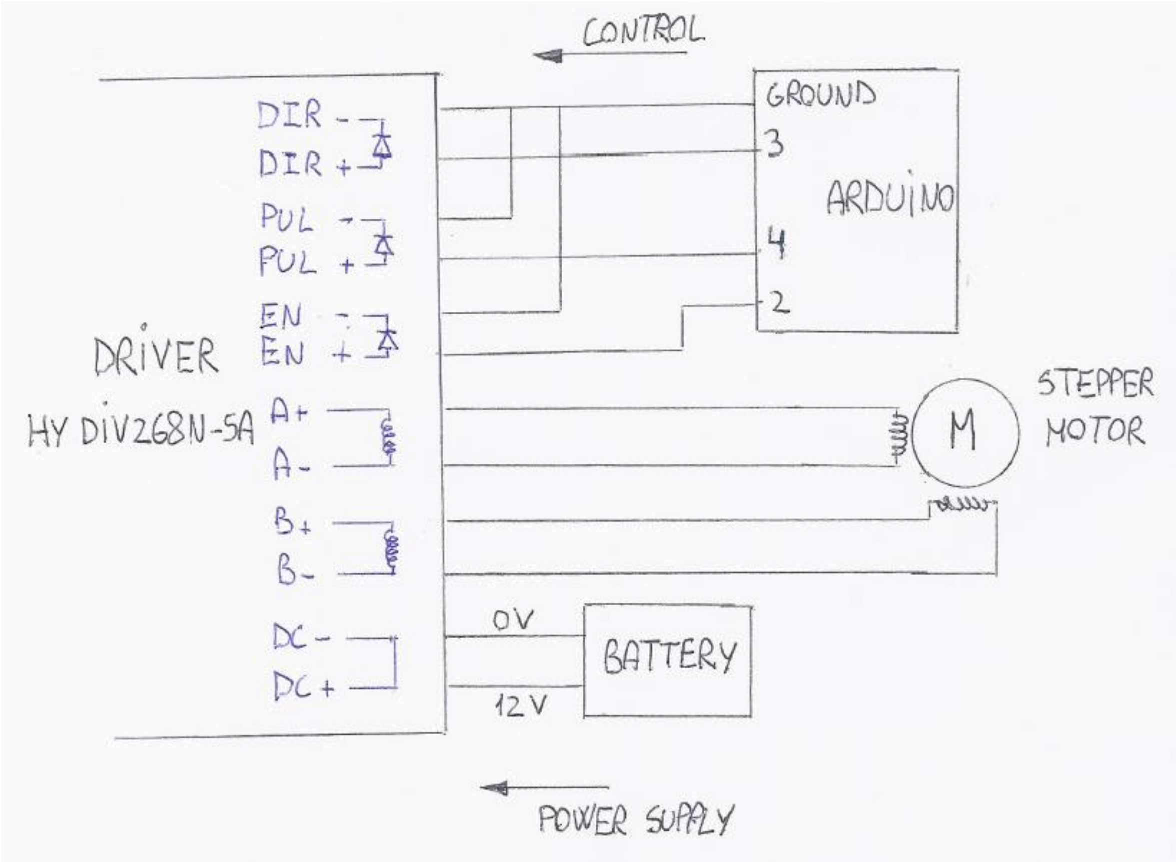


FIGURE 58: STEPPER MOTOR ELECTRONIC CONNECTION.

Furthermore, this driver has six switches that allow varying two interesting characteristics of the stepper motors. On the one hand, three of them are set up to allow varying the current of the motor supply. These changes of the current only affect to the torque given by the motor. Hence, as I want the maximum torque, I set up the driver to give the maximum intensity (5A). It has to be remarked that, depending on the working point of the mechanism, maybe lower currents are enough to move the cylinder. In this case, I would diminish the current, because surpassing the motor nominal current during a long while could lead to its own burning. On the other hand, there is a possibility, given by the other three switches, of increasing even more the number of steps per revolution of the stepper more. Actually, it divides each step into several micro steps, which allow controlling more precisely the position of the motor. Nevertheless, it also has a drawback, which is the fact that the more a step is divided, the lower torque a micro step provides. Therefore, as this design doesn´t need that precision but it does need the whole motor torque, I decline the idea of using micro step controlling.

Finally, considering that the motor makes 200 steps each revolution, I make a program to control stepper motor velocity, and accordingly, the cylinder velocity. This Arduino program is attached at the end of the document. This program allows introducing the desired cylinder frequency. Then, it calculates the time between two successive coil excitations as it is done below for a arbitrary frequency of 2 Hz, knowing that the number of steps of the motor is 200:

$$2\ Hz \rightarrow 2\frac{rev}{s} \rightarrow 0.5\frac{s}{rev}$$

$$t = 0.5\frac{s}{rev} * \frac{1rev}{200steps} = 0.0025\frac{s}{step}$$

t = time between two successive coil excitations

Accordingly, this program varies the coils excitation frequency depending on the cylinder frequency. It continues rotating at this frequency until the letter ´x´ is pressed on the keyboard. When this is done, the motor stops and the program ask for another cylinder frequency.

7.2.3. Encoder

As I have already said, I do use neither sensors nor close-loop control in this project. However, utilizing a close-loop control would be much more accurate. Thus, as a piece of advice for people working on this in the future, I would enhance the project using an encoder as a sensor of the position of the stepper motor.

An encoder is an electro mechanic device which allows measuring the velocity of any rotating axis. It consists of a disc rotating solidary with the axis (in this case, the motor), and a metallic sheet which has been scratched so that there are some holes in its surface. The disc contains some metallic contacts that can be touching the metallic sheet or introduced in the holes. These holes are designed in a way that makes every angle of the circle creating a different binary code. There is also an electric source connected with the sheet so that each contact between it and the metallic contactors produces an electric signal. The electronic components of the sheet can collect these electric signals and obtain the position of the axis thanks to the binary code associated with each angle.

As there is not enough physically space in the device, only some angle branches can be measured. Generally, if there is n contacts, the number of different measured positions of the axis is 2^n. Thus, this finite number of positions implies problems in the changes between two branches. As the data is based on mechanic contacts, it can be momentarily defective. I mean that, when the signals of the contacts are changing between two different binary codes, it is possible that the contacts don't chance at the same time so that fake binary codes may be given during the change, confusing the electronics. Hence, encoders don't usually use the normal binary code but a different one called Gray Code. This code has the special feature that only one position change while passing between two adjacent codes, which makes errors impossible to appear.

Regarding to whether they can calculate the exact position of the axis or not, there are two kinds of encoders: absolute encoders, which do permit knowing the exact angle of the axis; and the incremental ones, which only give the rotated angle. As I only need to control the velocity of the motor and I don't mind what its exact position is, it is enough, in this case, using an incremental encoder.

8. Summary

In this project, an experimental set up for the study of water VIV has been designed and built. The objective of the plant is achieving a continuous water flow passing a forcedly oscillating cylinder, provoking vortex shedding. In order to achieve it, the set-up has been designed consisting of four parts: a water closed circuit, a propel system, an external structure and a driving mechanism for the cylinder.

The water circuit has been built by joining two different parts: a rectangular duct and a pipes system. Regarding at the first one, its dimensions are 0.204 m width x 1 meter long x 0.25 meters high and it has been made by gluing PVC plates. The thickness of the plates and the width of the glued joints, have been made so that the flow tank is completely watertight. It also contains two transparent PVC plates which allow the view of the vortex shedding. Respecting to the pipe system, it consist on a long straight pipe and two 180 ˚ curves and it has been made with the bigger diameter pipes that was possible, to avoid head losses and turbulences on the rectangular duct. Thus, the selected pipes are 144.6 mm inside diameter PVC pipes, and the curves have a curvature radius of 89mm. All of the circuit is supported by an aluminium structure, made with standardized profiles. This structure protects the PVC plates and the joints between them against the bending moment that is going to appear due to the pressure of the water. This protection is absolutely necessary because, as I have said before, the glue joints cannot withstand this kind of load, as it focuses its effect on the edge of the glued join.

After looking deeply for a suitable pump without success, I decided that the water was going to be moved finally four 75 mm water propellers, two of them spinning clockwise and the other two spinning counter-clockwise, to avoid the rotational moment on the water. For supporting these propellers I have done a fluid dynamic structure, which opposes little resistance to the water flow. The propellers are going to be moved with two brushless motors and a transmission system consisting of timing belts and pulleys.

Regarding at the driving mechanism, finally, it consist of a rod mechanism, which allowing the transmission of the motor movement, and two linear axis, which permitting transforming this motion in a straight one. Linear and angular bearings have been used in order to minimize friction and I have selected a stepper motor to move this mechanism, as it ensures precise movements and a high torque for low motor velocities (in this case, the velocity range is between 60 to 180 rpm). In order to select the correct stepper motor I have also estimated the forces acting on the cylinder and the motor torque they required.

In addition, I have made two different automatic controls to drive both the brushless and the stepper motors. These controls have been done using Arduino and a electronic converter, in the case of the brushless motor, or a stepper motor driver, in the case of the stepper motor. As a result, I have two programs that allow me set the vortex shedding frequency and cylinder frequency respectively, and to stop the motor whenever I want.

8. Discussion and conclusions.

Regarding at my particular case, there have been so many external aspects that have affected to my practical work, which I am going to explain below at the same time that I explain the results of my bachelor thesis:

- I have successfully implemented the driving mechanism I describe in this theoretical report and I can control it as I have explained. Several tests have been done trying it in the air or inside the water and it has been seen that the device functions correctly for cylinder frequencies between 0.5 or 3 Hz. In theory, the cylinder should have been able to move at a range of frequencies between 0 and 3 Hz. However, practically it can be observed that at low velocities, the motor torque is not able to defeat the static friction and it doesn´t start spinning. Talking about friction, it has to be noticed that the friction of the linear bearings should have been take into account on the theoretical calculations, as its behavior is not as smooth as it was supposed to be, even greasing the axis where they are sliding.

- I have successfully implemented the propellers structure, as it was designed. Actually, I haven´t done complete test on the performance of device because there was a component (timing belts that have just arrived) but I have proved controlling one propeller connected to the motor with a round belt and it worked perfectly, so I hope that by the day of the presentation I will have achieved it. The only issue that may happen is that the propellers axes bend together due to the tension of the timing belts, provoking that the propellers hit to themselves. I have already thought about this, and if this happens, I will put a plate with holes inside the vertical threaded bars just above the upper propellers, so that it contributes to increase the rigidity of the structure.
  I have proved that the control of the motors works perfectly. However, as I have said before, the relation between the signals send by Arduino and the spinning velocity of the propellers and the flow velocity they cause, needs to be experimentally tested in the flow tank. Therefore, the introduced frequency on the program is not going to be real. Some trials are needed to obtain these relations and adjust the command in the Arduino program. I haven´t been able to do it because I haven´t finished the flow tank yet.

- The construction of the water circuit is still being done because of the late arriving of the ordered PVC plates (9 June). Nevertheless, most of it has already been done: the rectangular duct has been built and the pipes have been glued, so that it only lasts joining both parts, which I hope to do before the presentation too. It has to be said that there has been a problem with the curved pipes. Due to friction, they couldn´t be glued correctly and there is difference of length of 1.75 cm between the length of the two 180˚ curve pipes. However, this error is fixable and I think I will be able to fix it before the presentation.

- Most of the frame has been done too. It hasn´t been completely done because it can´t be closed before finishing the flow tank and placing it inside the structure.

- Two wood boxes have been done in order to harbor the electronics of the driving mechanism and the propellers structure.

Once arriving to this point, I have to said that it hasn´t been easy to carry out this project because it is difficult to adjust a theoretical model to the reality. In the real life, it is not possible to get all the materials you need or get them in the shapes you need, so the theoretical design has to be changed many times. Besides, in the reality, you cannot build everything you want. There exist knowledge, equipment and economic frontiers that slow you down. In addition, in the real life pieces doesn´t fit exactly as in the design due to errors or tolerances. Therefore, I have realized the extra-difficulty present in practical tasks, which I have never done in my home university.

Besides, it has been really difficult the starting of this project as I didn´t know exactly how to do design that was able to be built, where to look for the pieces I need, and who to ask my questions about particular matters (as I didn´t know the professors of the university). However, when Lieven Standaert has started helping me, the project has started to take form. Thus, it would have been

impossible doing this project without him, and I want to thank him for all the pieces of advice he gave me.

9. Bibliography

[1] C. T. Crowe, D. F. Elger, B. C. Williams, and J. A. Roberson, *Engineering Fluid Mechanics Ninth Edition*, vol. 1. 2015.

[2] Blevins, “Flow Induced Vibrations.” Krieger Publishing Co., Florida., 1990.

[3] John D Anderson, “Ludwig Prandtl ’ s Boundary Layer,” *Phys. Today*, no. December, p. 7, 2005.

[4] R. F. Gonzalez, “Modelacion de flujo laminar y transferencia de calor en haz de tubos,” 2012.

[5] *Vortex shedding from a circular cylinder*. .

[6] H. Blackburn and R. Henderson, “Lock-In Behaviour in Simulated Vortex-Induced Vibration,” *Experimental Thermal and Fluid Science*, vol. 12. pp. 184–189, 1996.

[7] R. Bourguet, G. E. Karniadakis, and M. S. Triantafyllou, “Multi-frequency vortex-induced vibrations of a long tensioned beam in linear and exponential shear flows,” *J. Fluids Struct.*, vol. 41, pp. 33–42, 2013.

[8] P. W. BEARMAN, “Understanding and predicting vortex-induced vibrations,” *J. Fluid Mech.*, vol. 634, p. 1, 2009.

[9] C. H. K. Williamson and R. Govardhan, “Vortex-Induced Vibrations,” *Annu. Rev. Fluid Mech.*, vol. 36, no. 1, pp. 413–455, 2004.

[10] M. Saarlas, “Aircraft Performance,” *Aircr. Perform.*, pp. 1–282, 2007.

[11] W. V. Titow, “‘PVC Technology Handbook,’” *Elsevier Appl. Sci. Publ.*

[12] UPC, “Diseño de Uniones Adhesivas,” no. 3, pp. 1–4.

[13] R. García Ledesma, “Diseño y comportamiento de uniones estructurales mecánicas y adhesivas. Condiciones superficiales y operacionales.,” 2013.

[14] J. Valencia, “Flujo Viscoso en Conductos. Pérdidas Primarias.,” *Apunt. Mecánica Fluidos*, pp. 98–114.

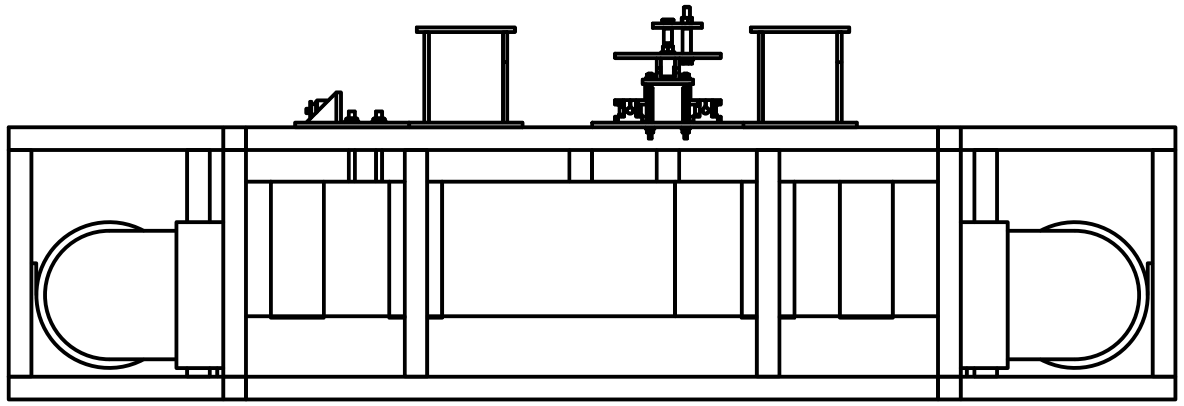
[15] J. Valencia, “Pérdidas Localizadas En Conductos,” .

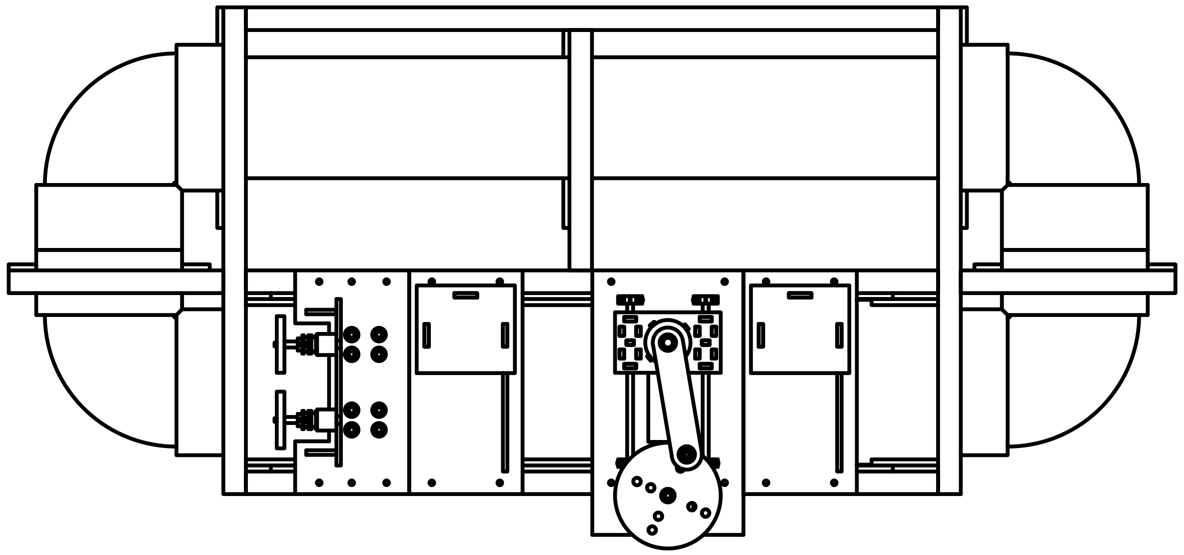
[16] C. R. C. Santiago, T. Chu, and K. H. Wang, “Study of the Head Loss Associated with a Fluid Flowing through a Porous Screen,” no. August, pp. 1–26, 2007.

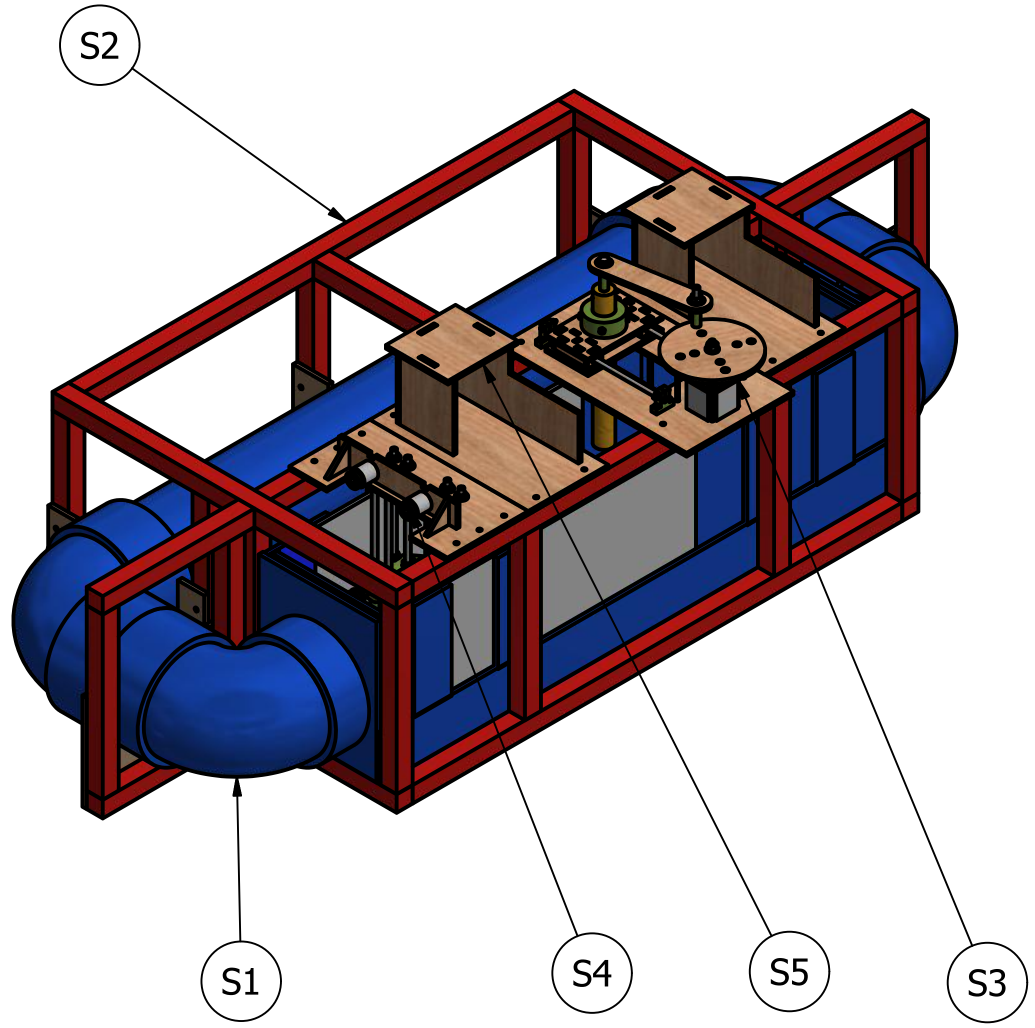
[17] “http://www.edur.com/en/02-Products/pdf/Projektierung/WA3_E.pdf.” .

[18] J. García, “CLASIFICACIÓN Y DESCRIPCIÓN DE MÁQUINAS HIDRÁULICAS,” 2012.

[19] C. Norberg, “Fluctuating lift on a circular cylinder: Review and new measurements,” *J. Fluids Struct.*, vol. 17, no. 1, pp. 57–96, 2003.



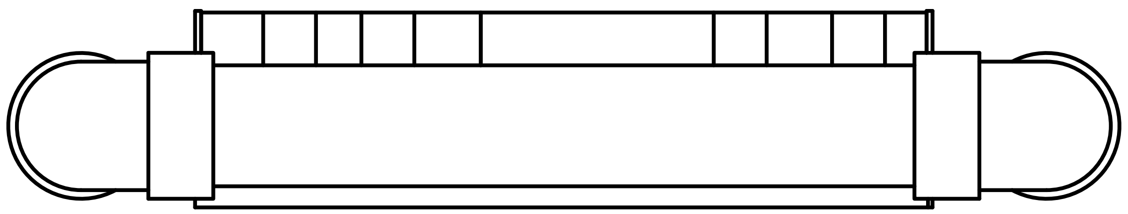


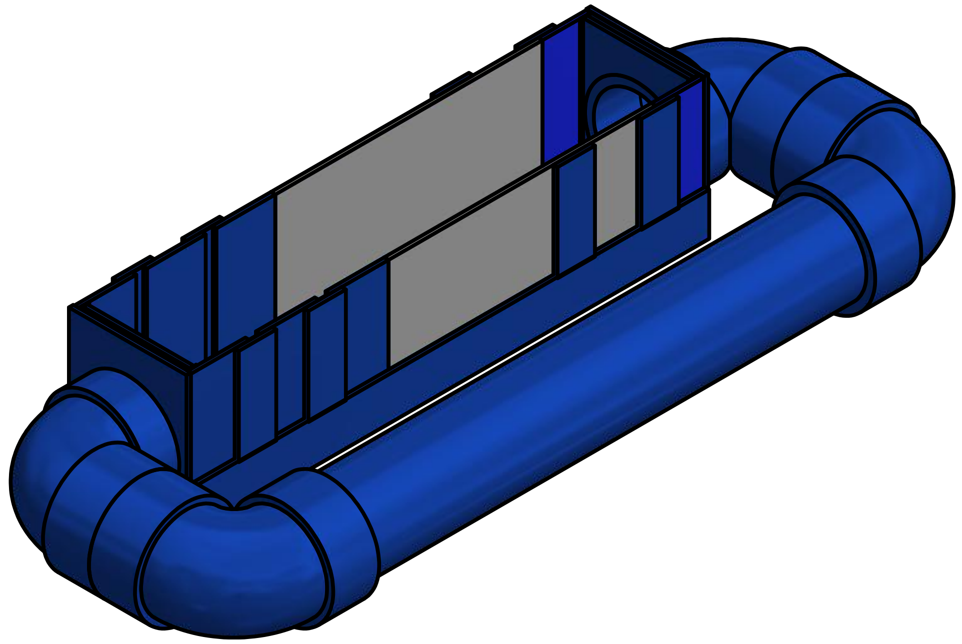


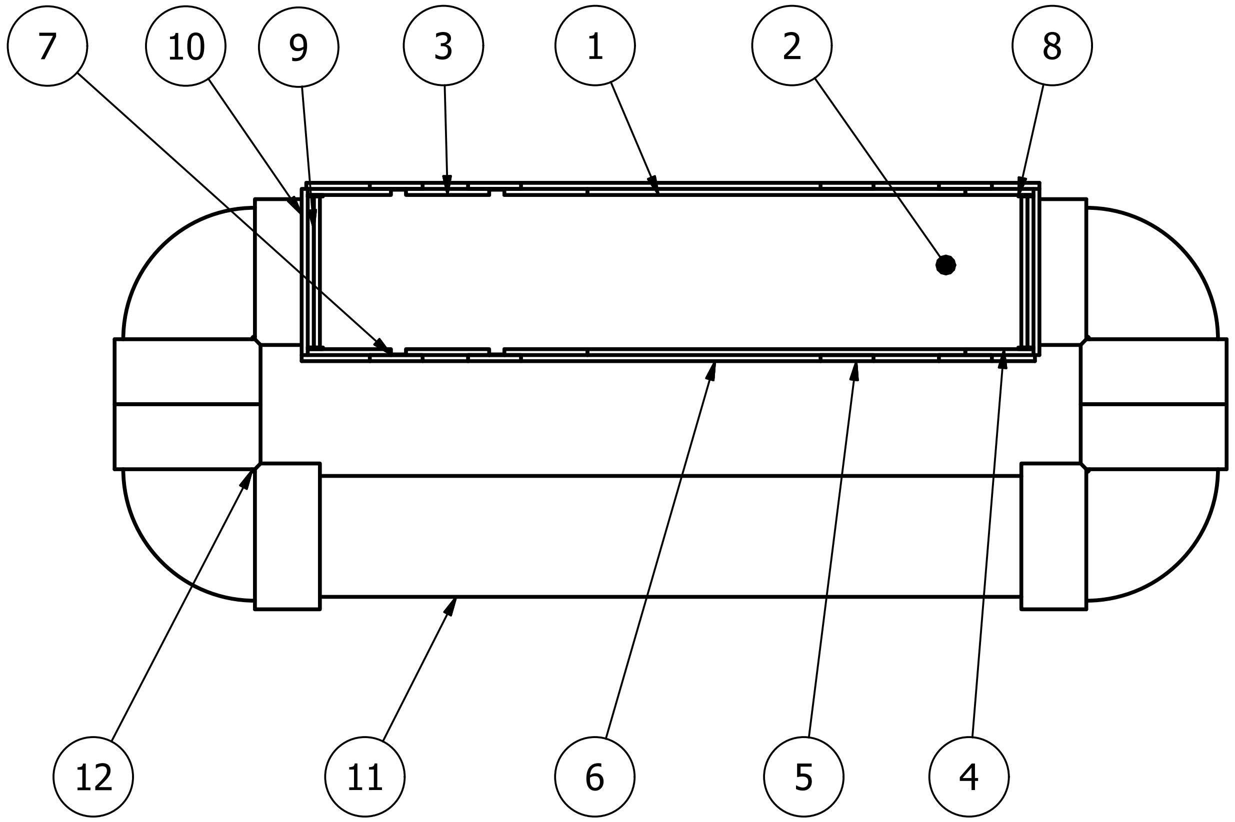
Parts	
Number	Subset
S1	Water Circuit (Flow Tank and Pipes)
S2	Frame
S3	Driving Mechanism
S4	Propellers Structure
S5	Electronics Support

Designed by	Checked by	Approved by	Date	Date	Scale: 1:10
mhermoso				2/05/2016	

V.I.V experimental set up	Miguel Hermoso de Mendoza Sarasa		
	V.I.V experimental set up	Edition	Sheet 1 / 1





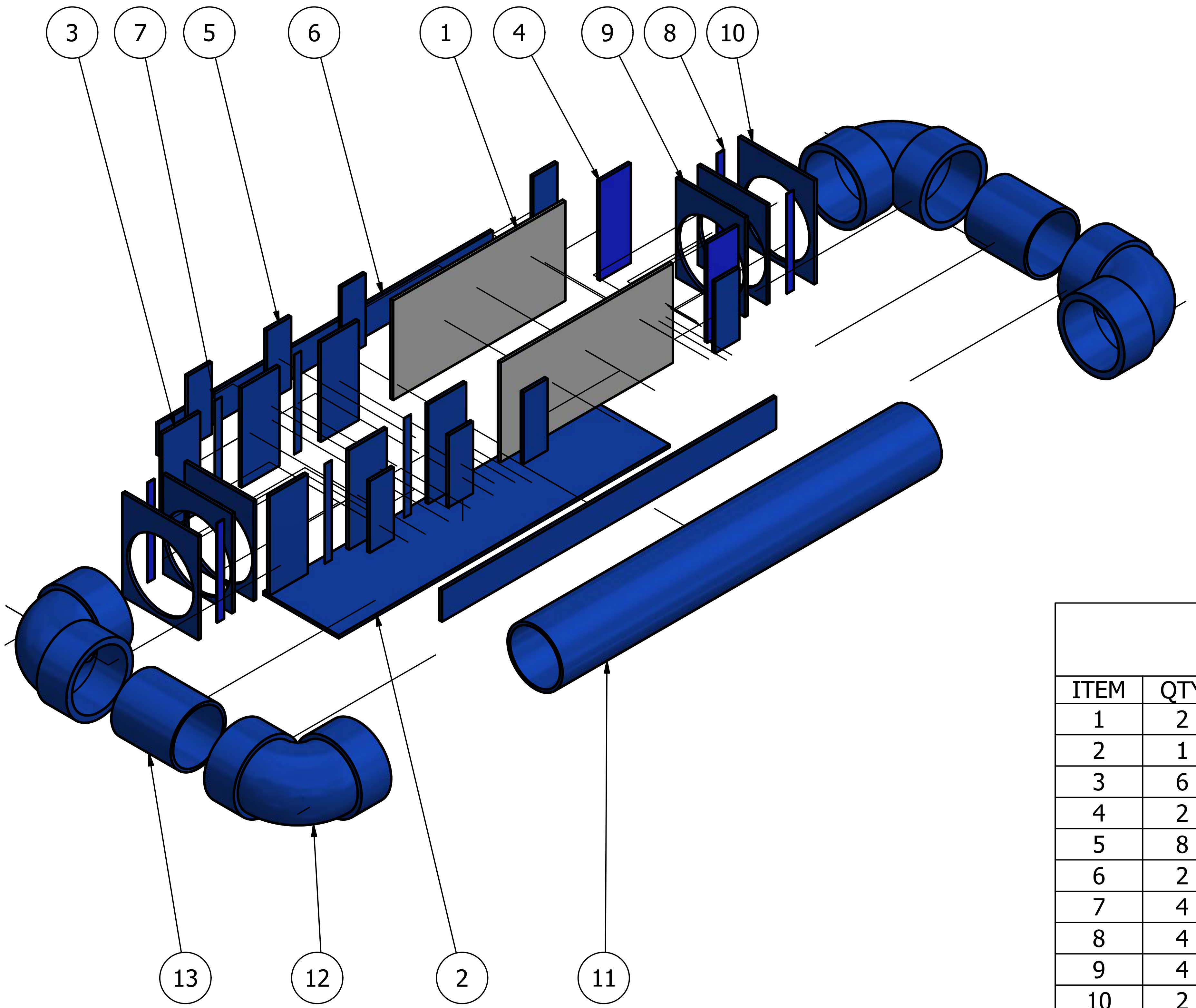


PARTS LIST

ITEM	QTY	PART NUMBER	MATERIAL
1	2	Plate 500 x 250 x 8 mm	Transparent PVC
2	1	Plate 960 x 220 x 8 mm	PVC
3	6	Plate 250 x 110 x 8 mm	PVC
4	2	Plate 250 x 90 x 8 mm	PVC
5	8	Plate 180 x 70 x 8 mm	PVC
6	2	Plate 970 x 80 x 8 mm	PVC
7	4	Plate 20 x 250 x 1 mm	PVC
8	4	Plate 20 x 250 x 2 mm	PVC
9	4	Inside Cap	PVC
10	2	Outside Cap	PVC
11	1	Tube D=160 mm, d=144.6 mm L=1100 mm	PVC
12	4	Curve Pipe d=160mm	PVC
13	2	Tube D=160 mm, d=144.6 mm L=172 mm	PVC

Designed by	Checked by	Approved by	Date	Date	Scale: 1:10
mhermoso				2/05/2016	

S1. Water Circuit	Miguel Hermoso de Mendoza Sarasa		
	V.I.V experimental set up	Edition	Sheet 1 / 7

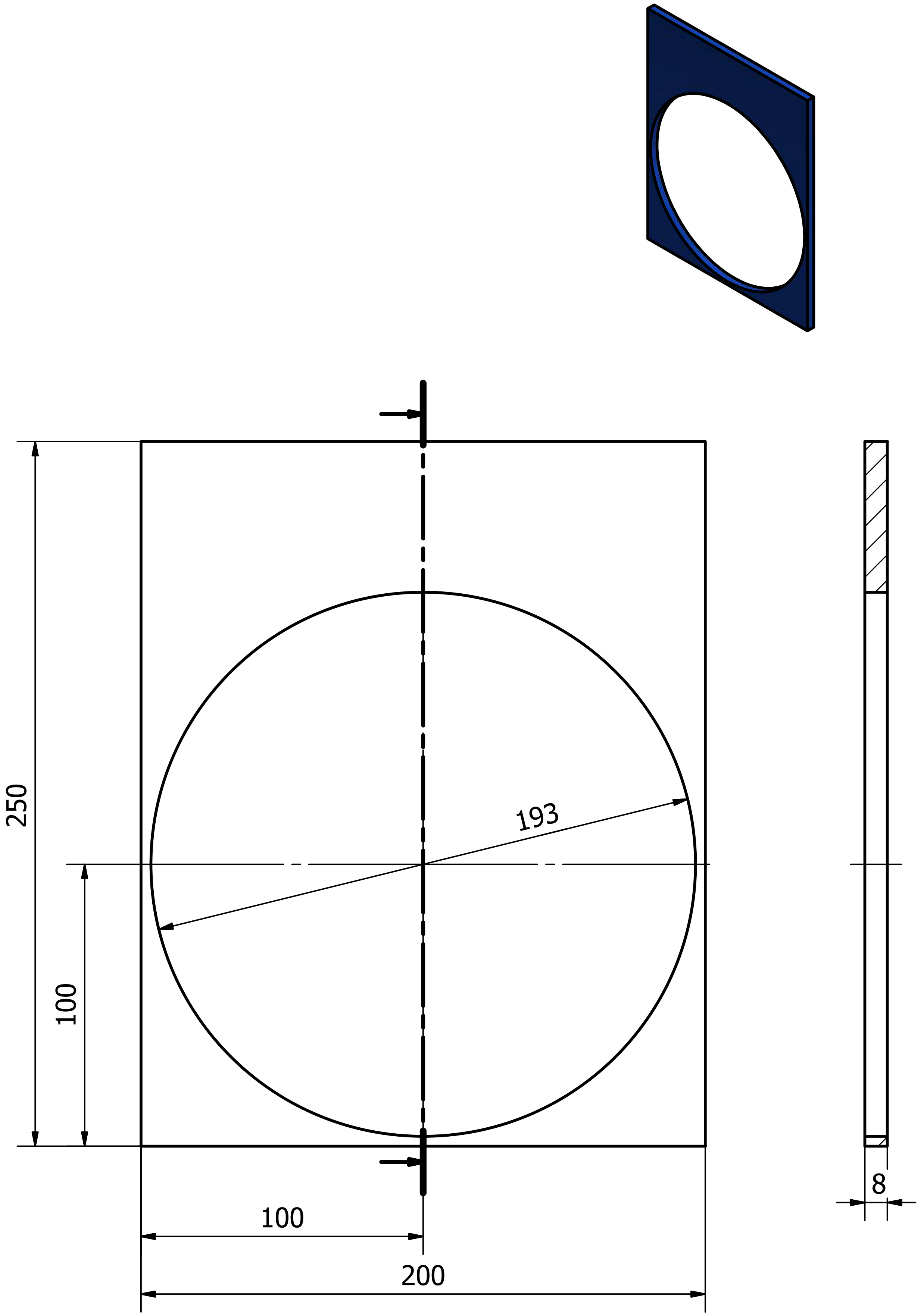


PARTS LIST

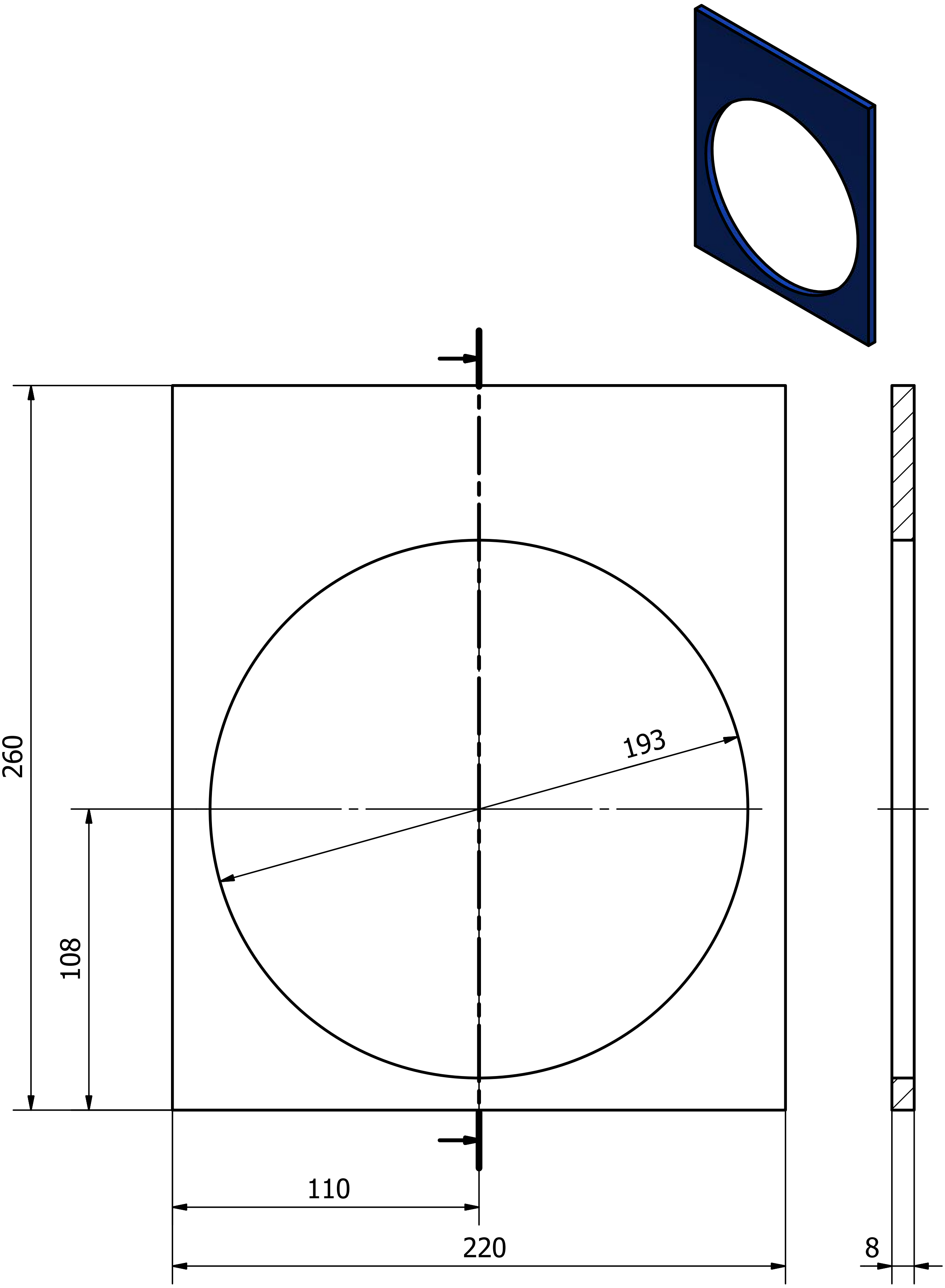
ITEM	QTY	PART NUMBER	MATERIAL
1	2	Plate 500 x 250 x 8 mm	Transparent PVC
2	1	Plate 960 x 220 x 8 mm	PVC
3	6	Plate 250 x 110 x 8 mm	PVC
4	2	Plate 250 x 90 x 8 mm	PVC
5	8	Plate 180 x 70 x 8 mm	PVC
6	2	Plate 970 x 80 x 8 mm	PVC
7	4	Plate 20 x 250 x 1 mm	PVC
8	4	Plate 20 x 250 x 2 mm	PVC
9	4	Inside Cap	PVC
10	2	Outside Cap	PVC
11	1	Tube D=160 mm, d=144.6 mm L=1100 mm	PVC
12	4	Curve Pipe d=160mm	PVC
13	2	Tube D=160 mm, d=144.6 mm L=172 mm	PVC

Designed by	Checked by	Approved by	Date	Date	Scale: 1:10
mhermoso				2/05/2016	

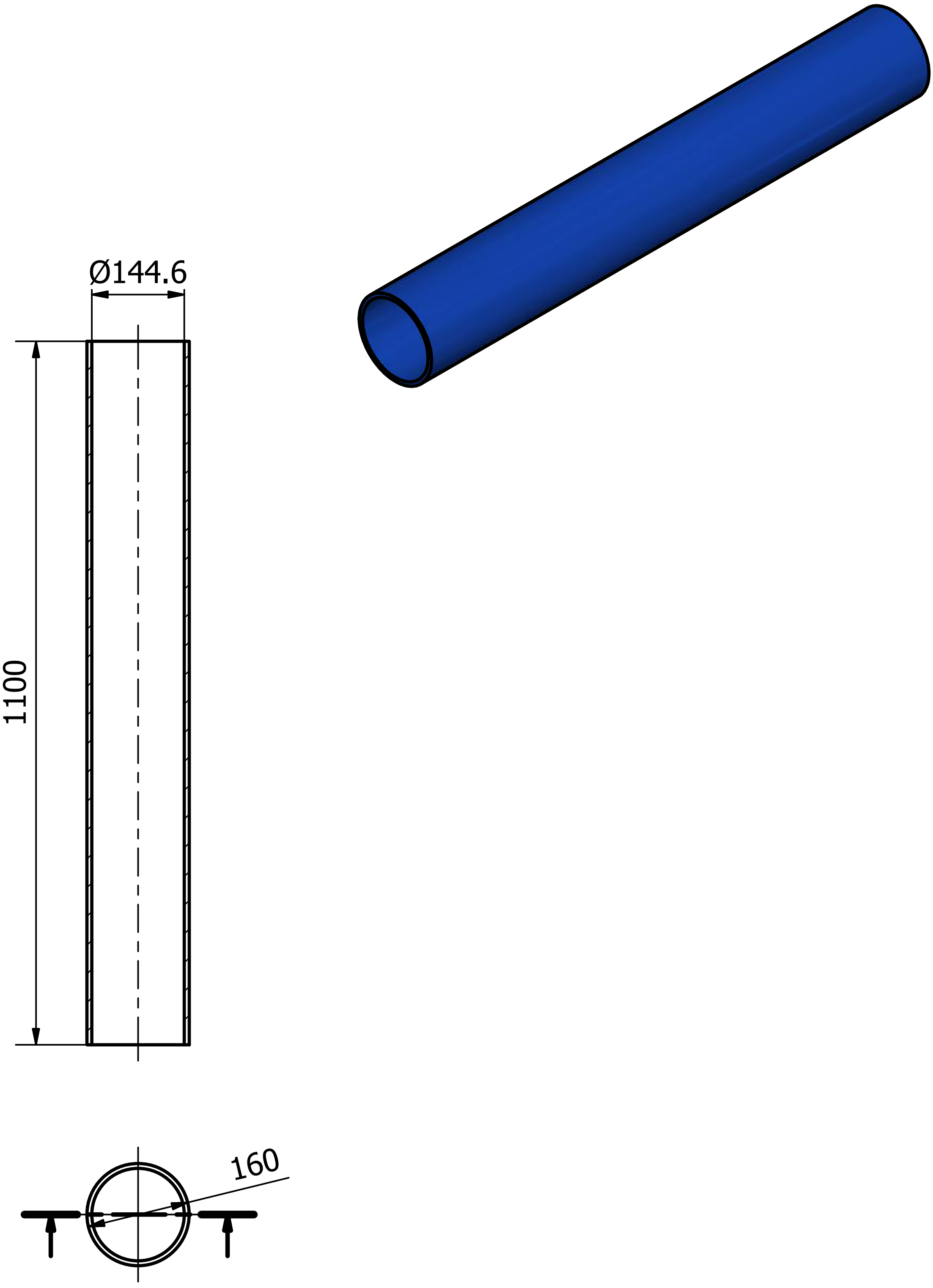
S1. Exploded Projection	Miguel Hermoso de Mendoza Sarasa		
	S1. Water Circuit	Edition	Sheet 2 / 7



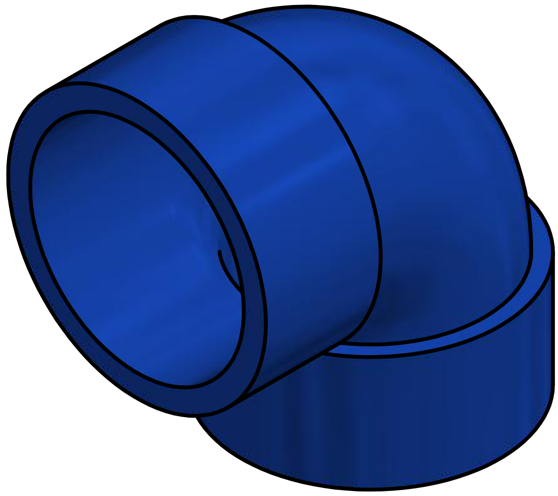
Designed by mhermoso	Checked by	Approved by	Date	Date 2/05/2016	Scale: 1:2
S1.9. Inside Cap	Miguel Hermoso de Mendoza Sarasa				
	S1. Water Circuit			Edition	Sheet 3 / 7

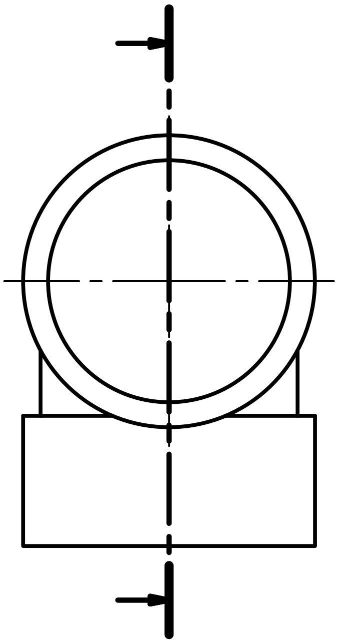


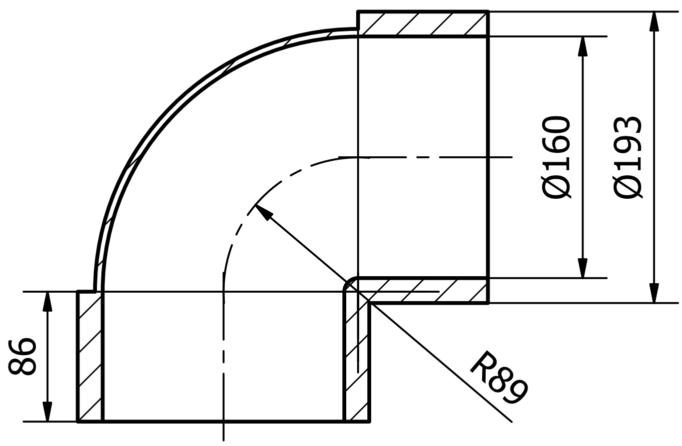
Designed by	Checked by	Approved by	Date		Date	Scale: 1:2
mhermoso					2/05/2016	
S1.10. Outside Cap			Miguel Hermoso de Mendoza Sarasa			
			S1. Water Circuit		Edition	Sheet 4 / 7



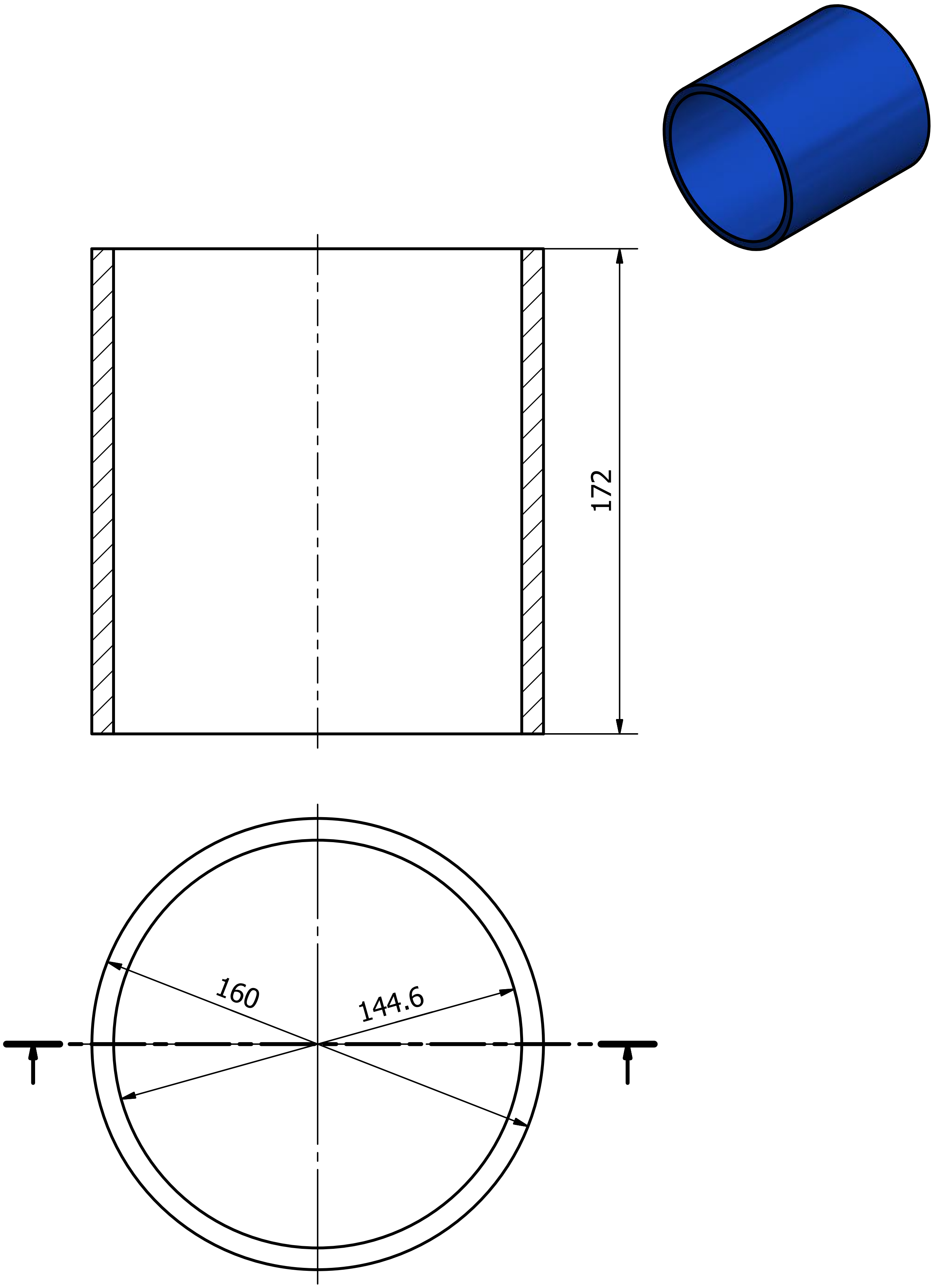
Designed by mhermoso	Checked by	Approved by	Date	Date 2/05/2016	Scale: 1:10
S1.11. Tube D=160 mm, d=144.6 mm L=1100 mm			Miguel Hermoso de Mendoza Sarasa		
			S1. Water Circuit	Edition	Sheet 5 / 7



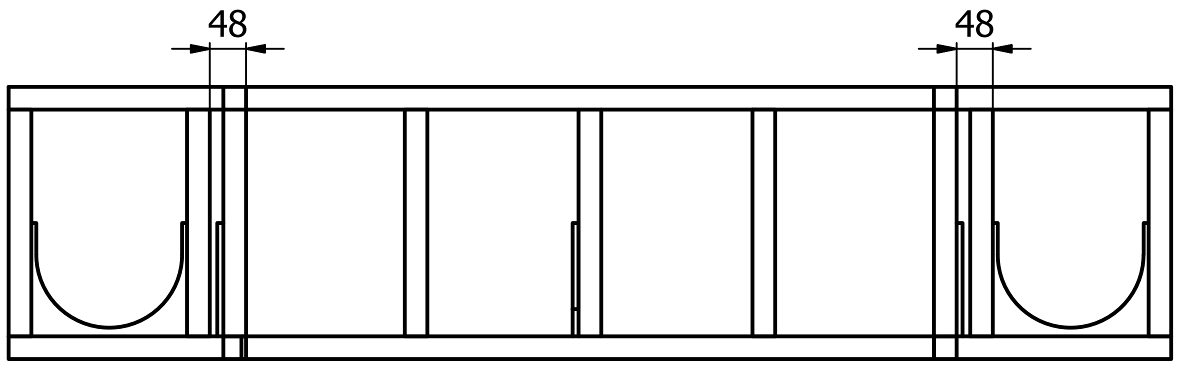


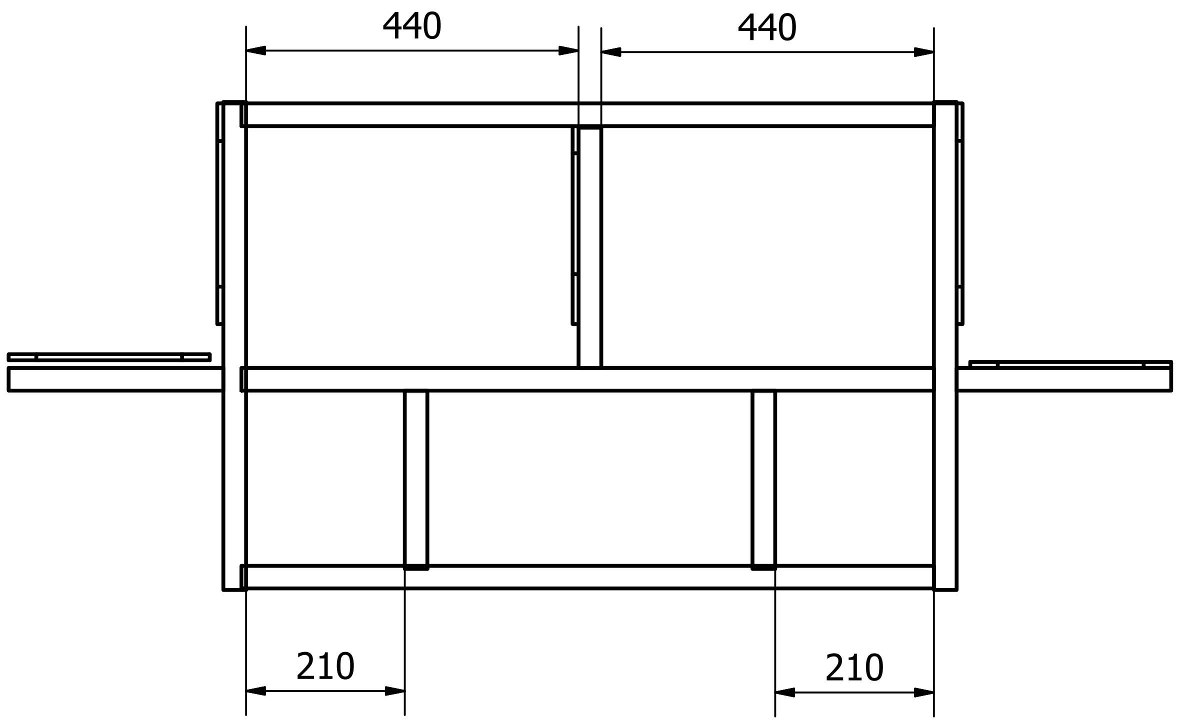


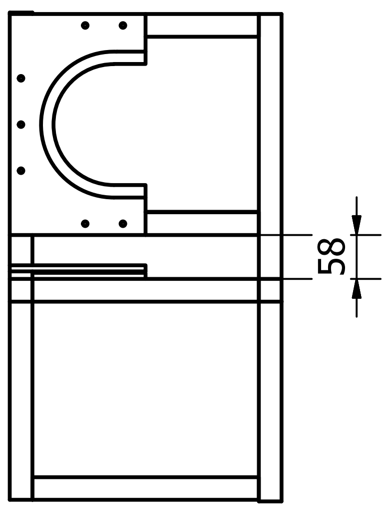
Designed by	Checked by	Approved by	Date	Date	Scale: 1:5
mhermoso				2/05/2016	
S1.12. Curve Pipe d=160mm			Miguel Hermoso de Mendoza Sarasa		
			S1. Water Circuit	Edition	Sheet 6 / 7

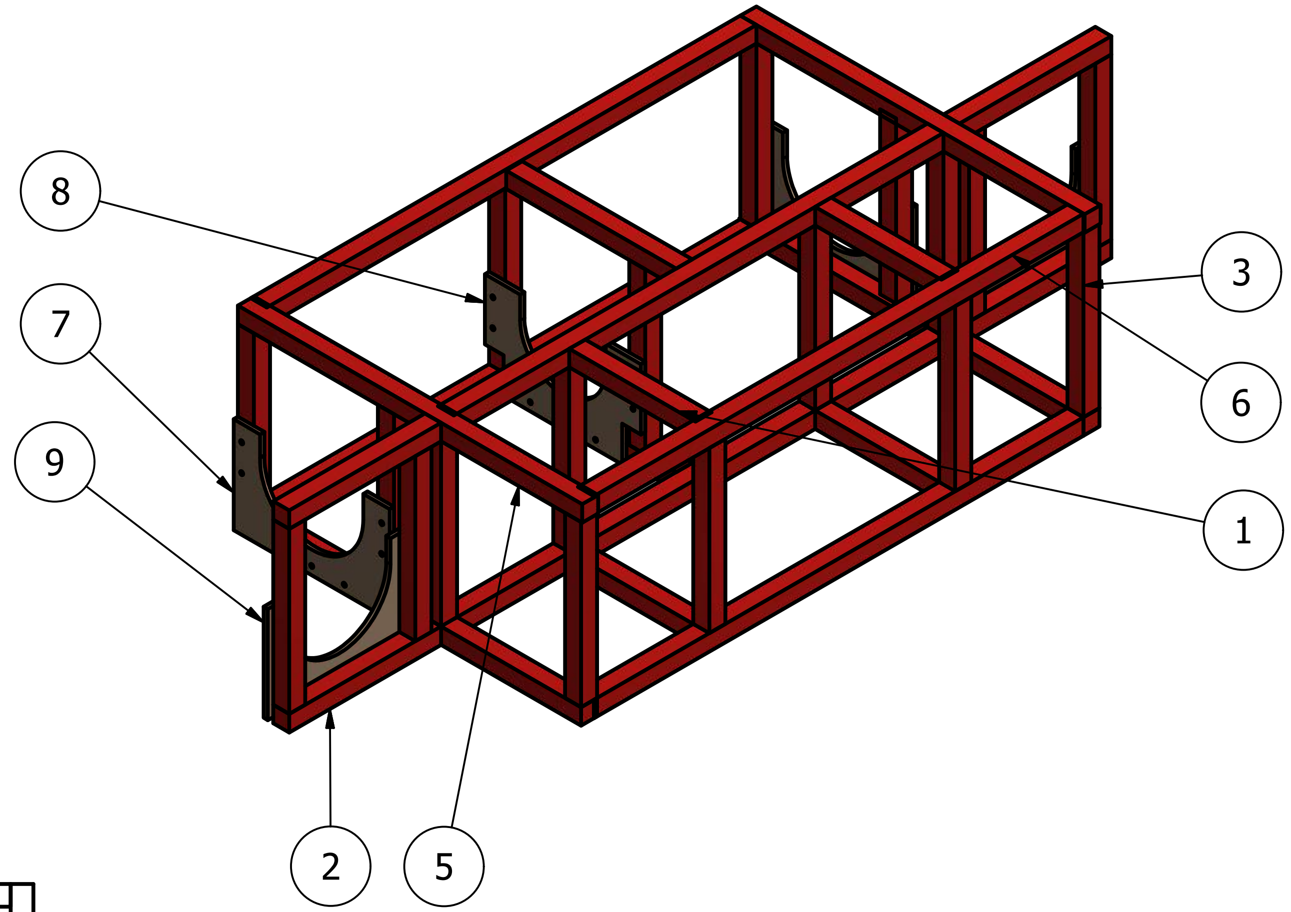


Designed by mhermoso	Checked by	Approved by	Date		Date 2/05/2016	Scale: 1:2
S1.13. Tube D=160 mm, d=144.6 mm L=172 mm			Miguel Hermoso de Mendoza Sarasa			
			S1. Water Circuit		Edition	Sheet 7 / 7





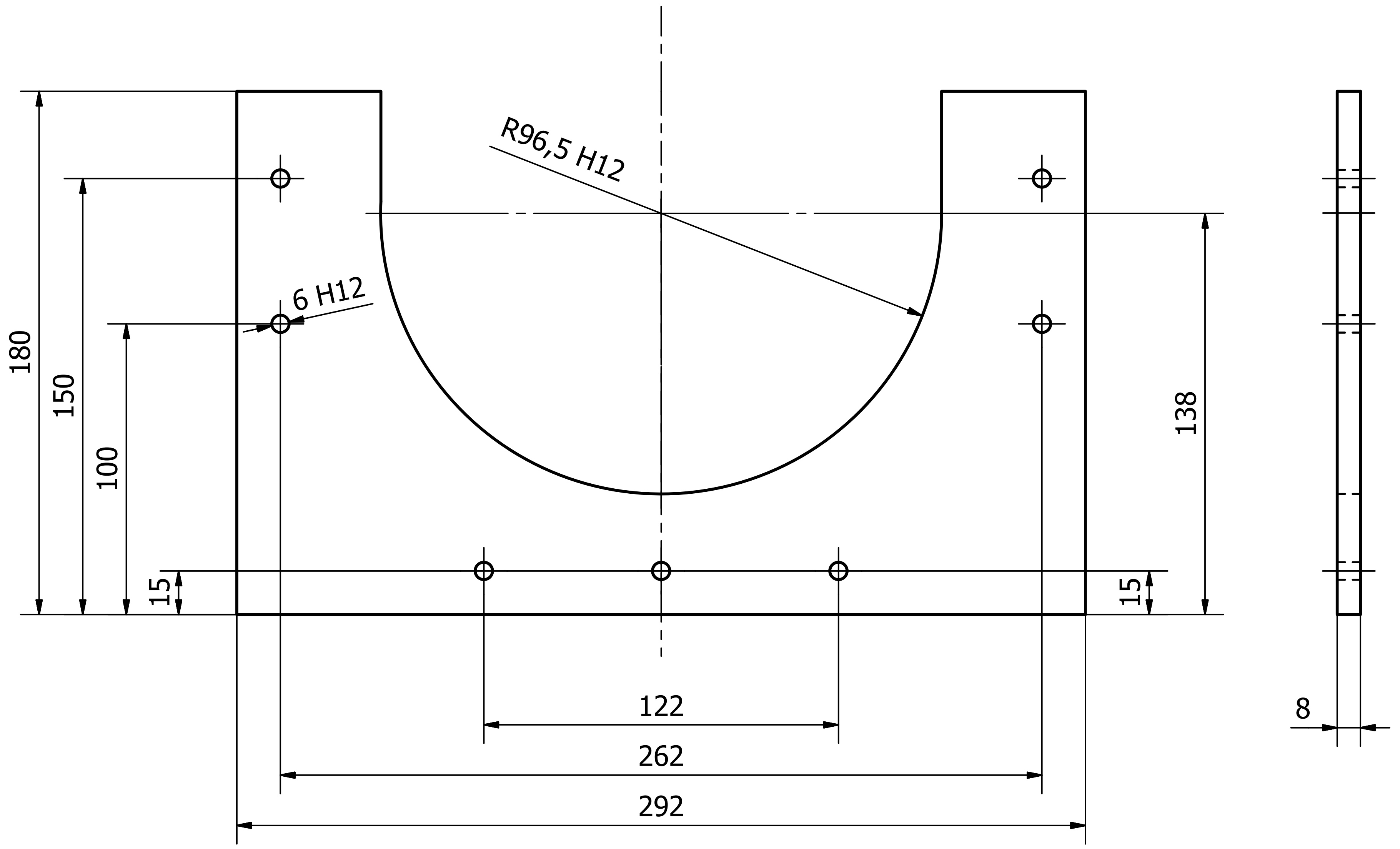


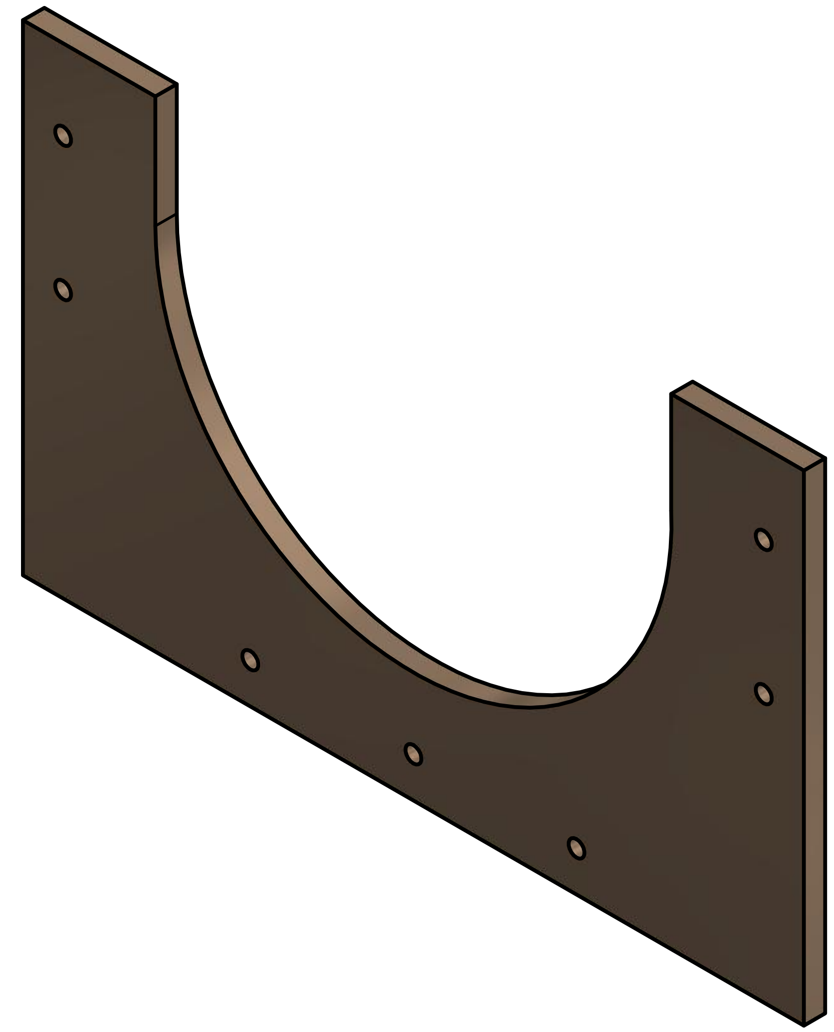


PARTS LIST			
ITEM	QTY	PART NUMBER	MATERIAL
1	4	Profile 30x30L I-Type Slot 6 length=236 mm	Aluminium
2	4	Profile 30x30L I-Type Slot 6 length=284 mm	Aluminium
3	18	Profile 30x30L I-Type Slot 6 length=300 mm	Aluminium
4	2	Profile 30x30L I-Type Slot 6 length=318 mm	Aluminium
5	4	Profile 30x30L I-Type Slot 6 length=644 mm	Aluminium
6	6	Profile 30x30L I-Type Slot 6 length=916 mm	Aluminium
7	2	Extreme Pipe Support	Burch 8mm
8	1	Central Pipe Support	Burch 8mm
9	2	Curv Pipe Support	Burch 8mm

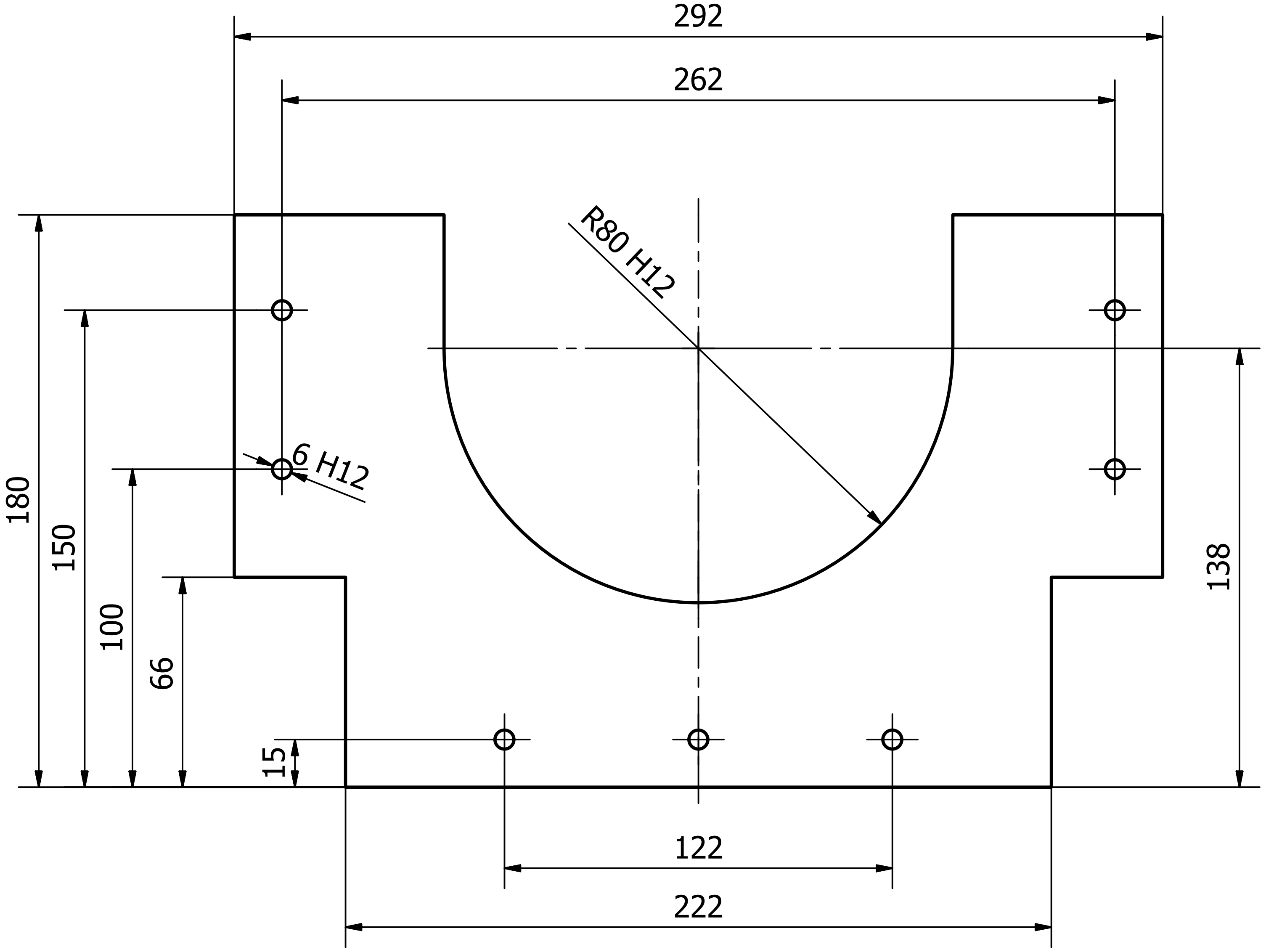
Designed by	Checked by	Approved by	Date	Date	Scale: 1:10
mhermoso				2/05/2016	

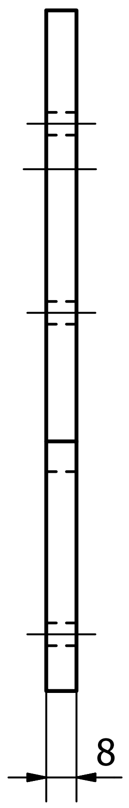
V.I.V experiment set up	Miguel Hermoso de Mendoza Sarasa		
	S2. Frame	Edition	Sheet 1 / 4

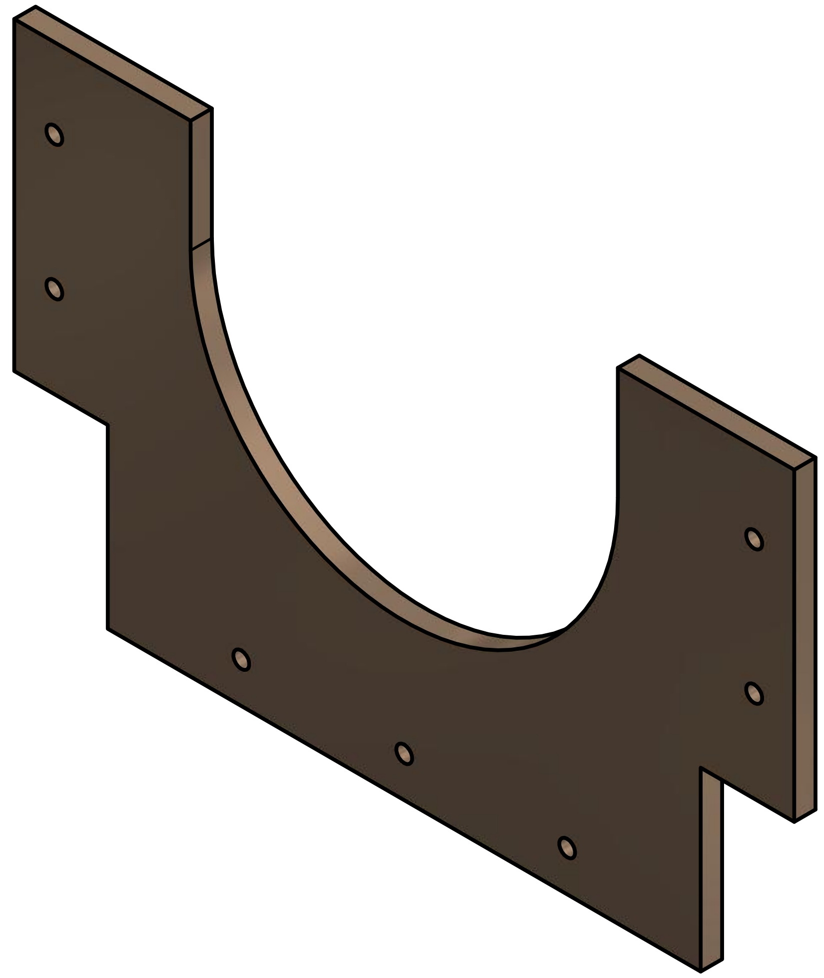




Designed by	Checked by	Approved by	Date		Date	Scale: 1:2
mhermoso					2/05/2016	
S2.7. Extreme Pipe Support		Miguel Hermoso de Mendoza Sarasa				
		S2. Frame			Edition	Sheet 2 / 4

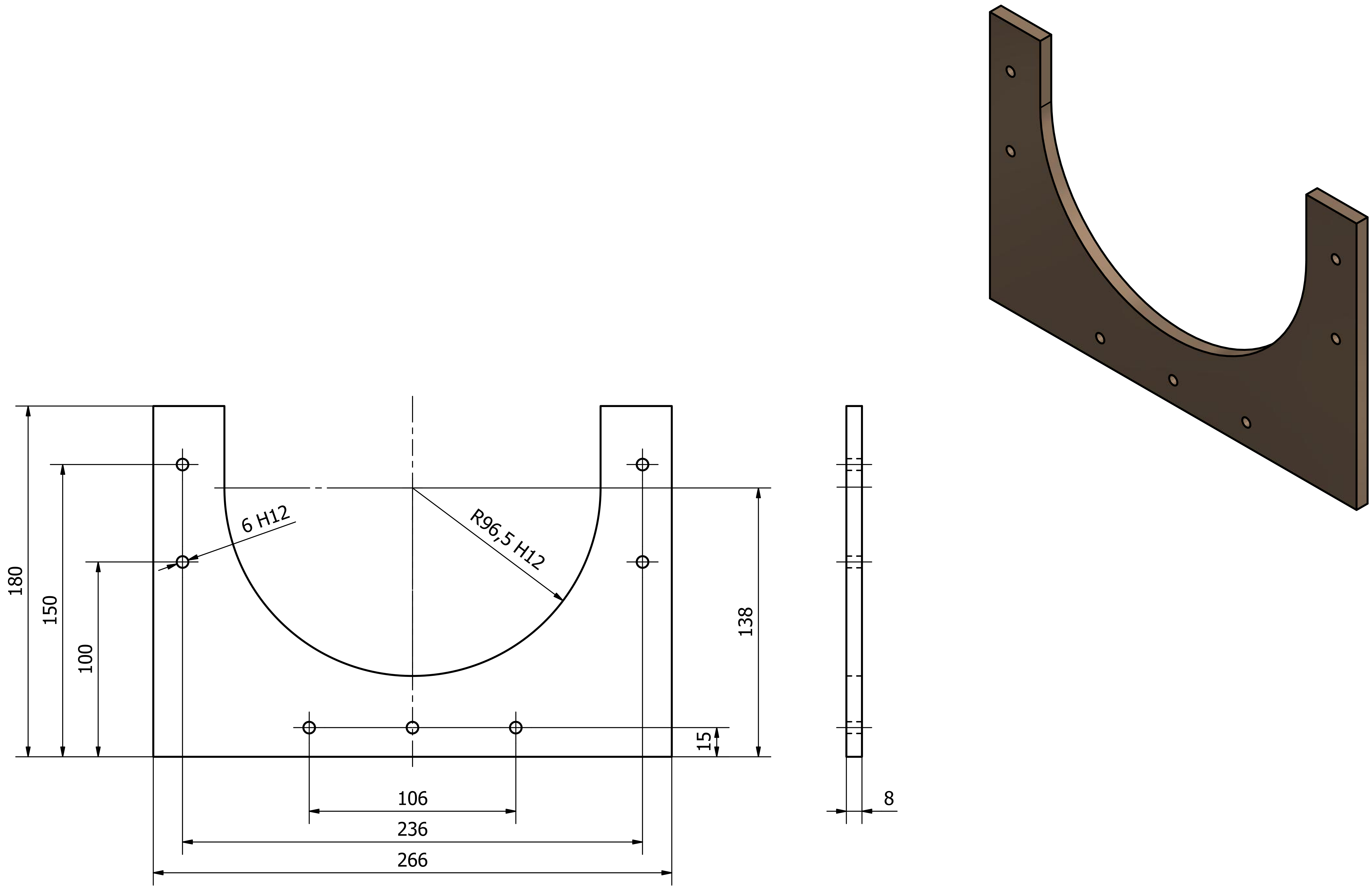




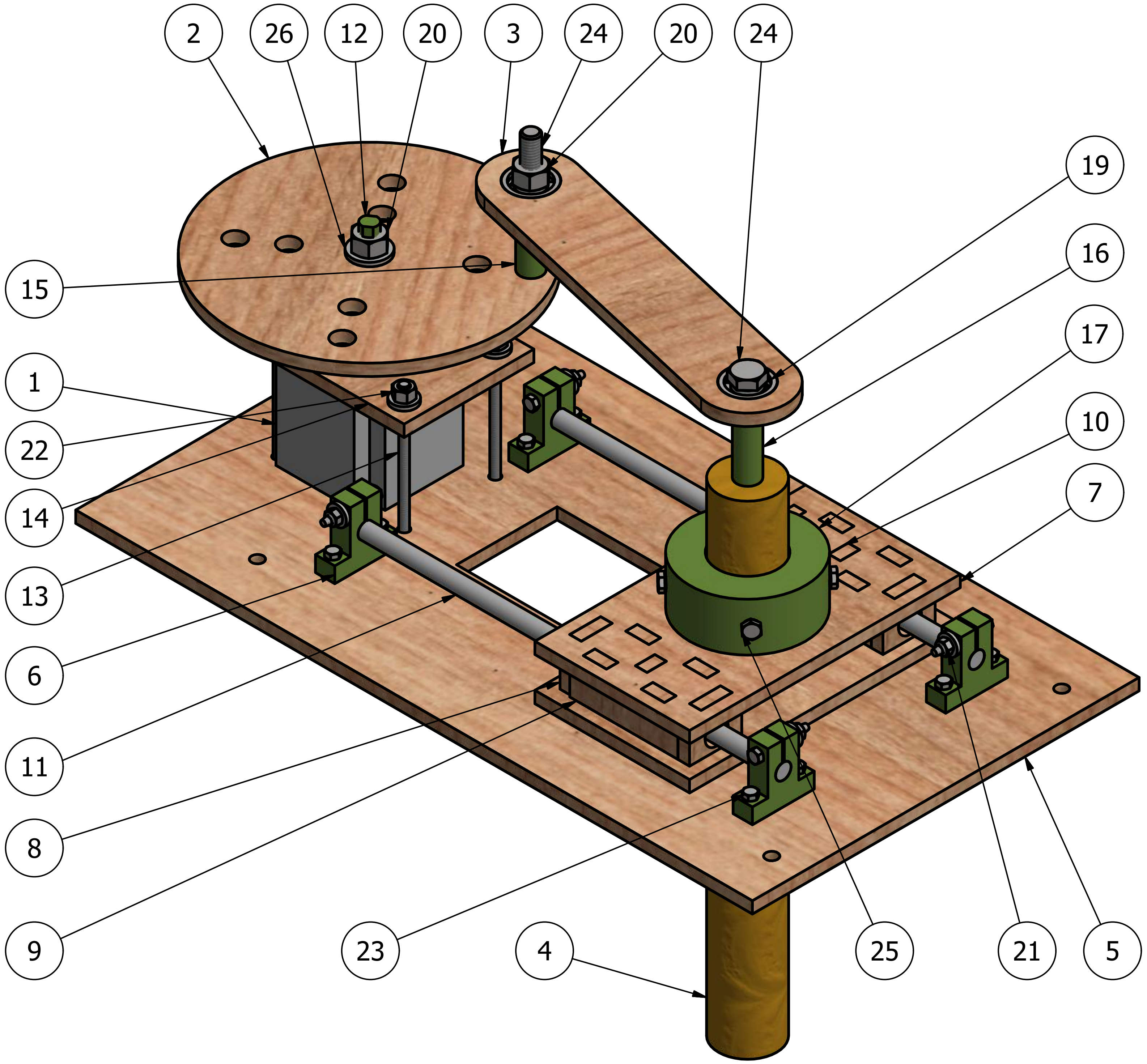


Designed by	Checked by	Approved by	Date	Date	Scale: 1:2
mhermoso				2/05/2016	

S2.8. Central Pipe Support	Miguel Hermoso de Mendoza Sarasa		
	S2. Frame	Edition	Sheet 3 / 4



Designed by	Checked by	Approved by	Date		Date	Scale: 1:2
mhermoso					2/05/2016	
S2.9. Lateral Pipe Support	Miguel Hermoso de Mendoza Sarasa					
	S2. Frame				Edition	Sheet 4 / 4

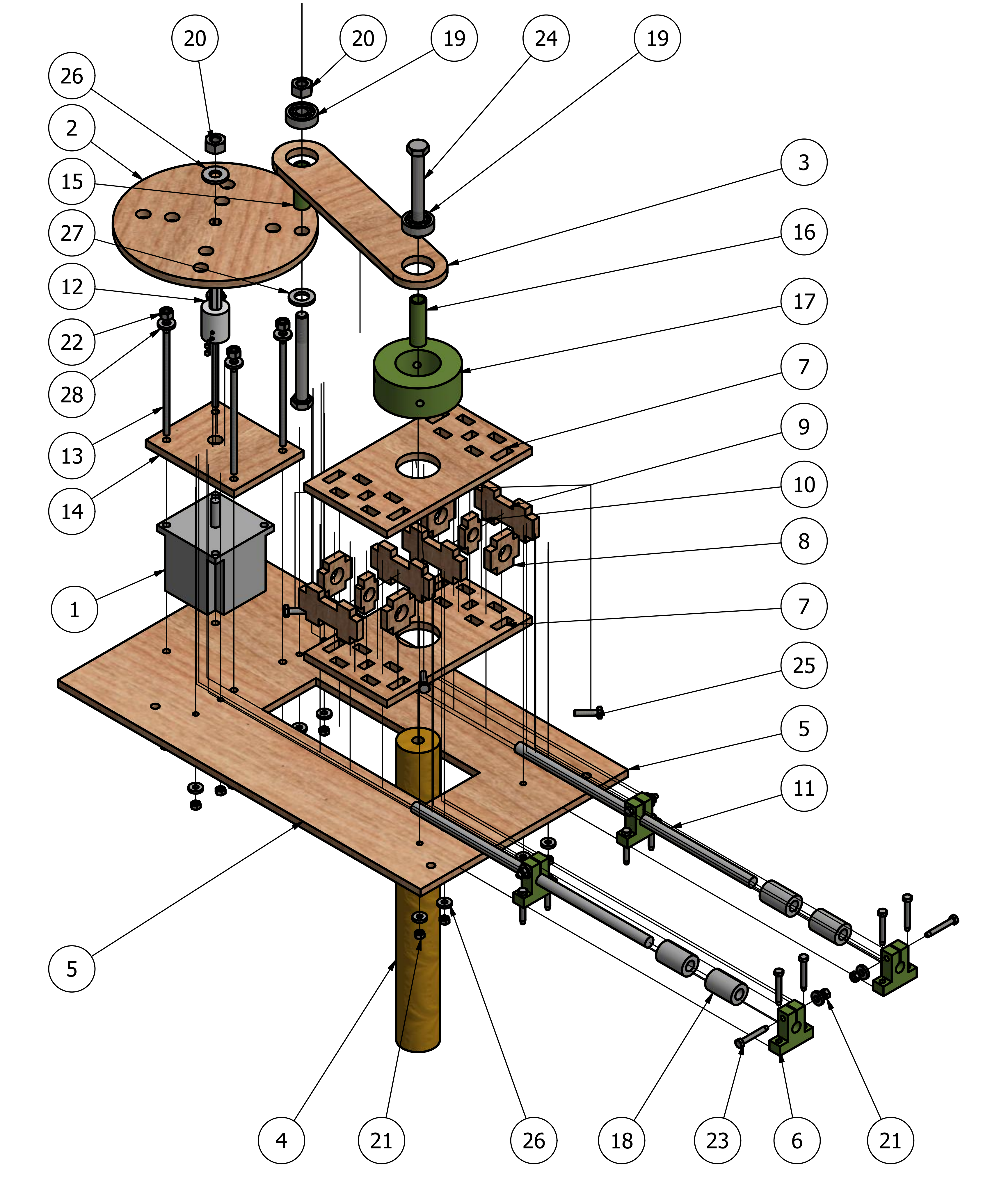


PARTS LIST

ITEM	QTY	PART NUMBER	DESCRIPTION	MATERIAL
1	1	Stepper Motor		Generic
2	1	Crank		Burch 6mm
3	1	Connecting Rod		Burch 6mm
4	1	Cylinder		Plastic
5	1	Guide		Burch 6mm
6	4	Support Axis		PLA
7	2	Box Plate		Burch 6mm
8	4	Box Piece 1		Burch 6mm
9	4	Box Piece 2		Burch 6mm
10	2	Box Piece 3		Burch 6mm
11	2	Bearings Axis		PLA
12	1	Transmisor		PLA
13	4	Vertical Threaded Axis		Steel
14	1	Fixing plate		Burch 6mm
15	1	Small Column		PLA
16	1	Big Column		PLA
17	1	Support Cylinder		PLA
18	4	Linear Bearing d=8mm		Steel
19	2	Rolling bearing B708 C GB/T 292-2007	Angular contact ball bearings d=8mm	Steel, Mild
20	2	ANSI B18.2.4.2M - M8x1.25	Metric Hex Nuts Styles 2	Steel, Mild
21	12	ANSI B18.2.4.2M - M4x0.6	Metric Hex Nuts Styles 2	Steel, Mild
22	8	ANSI B18.2.4.2M - M5x0.8	Metric Hex Nuts Styles 2	Steel, Mild
23	12	IFI 502 - M4x0.6 x 25	Hex Head Tapping Screw - Metric	Steel, Mild
24	2	ISO 4014 - M8 x 70	Hex-Head Bolt	Stainless Steel, 440C
25	4	AS 1110 - M5 x 16	ISO metric hexagon precision screws	Steel, Mild
26	1	Washer d=8mm		Generic
27	1	Washer d=10mm		Generic
28	20	Washer d=4mm		Generic
29	4	AS 1110 - M2 x 6	ISO metric hexagon precision screws	Steel, Mild

Designed by	Checked by	Approved by	Date	Date	Scale: 1:2
mhermoso				2/05/2016	

S3. Driving Mechanism	Miguel Hermoso de Mendoza Sarasa		
	V.I.V experimental set up	Edition	Sheet 1 / 17

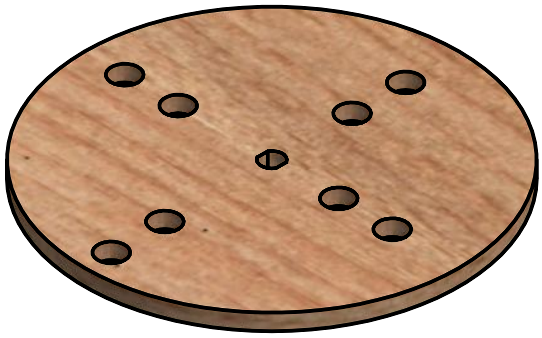


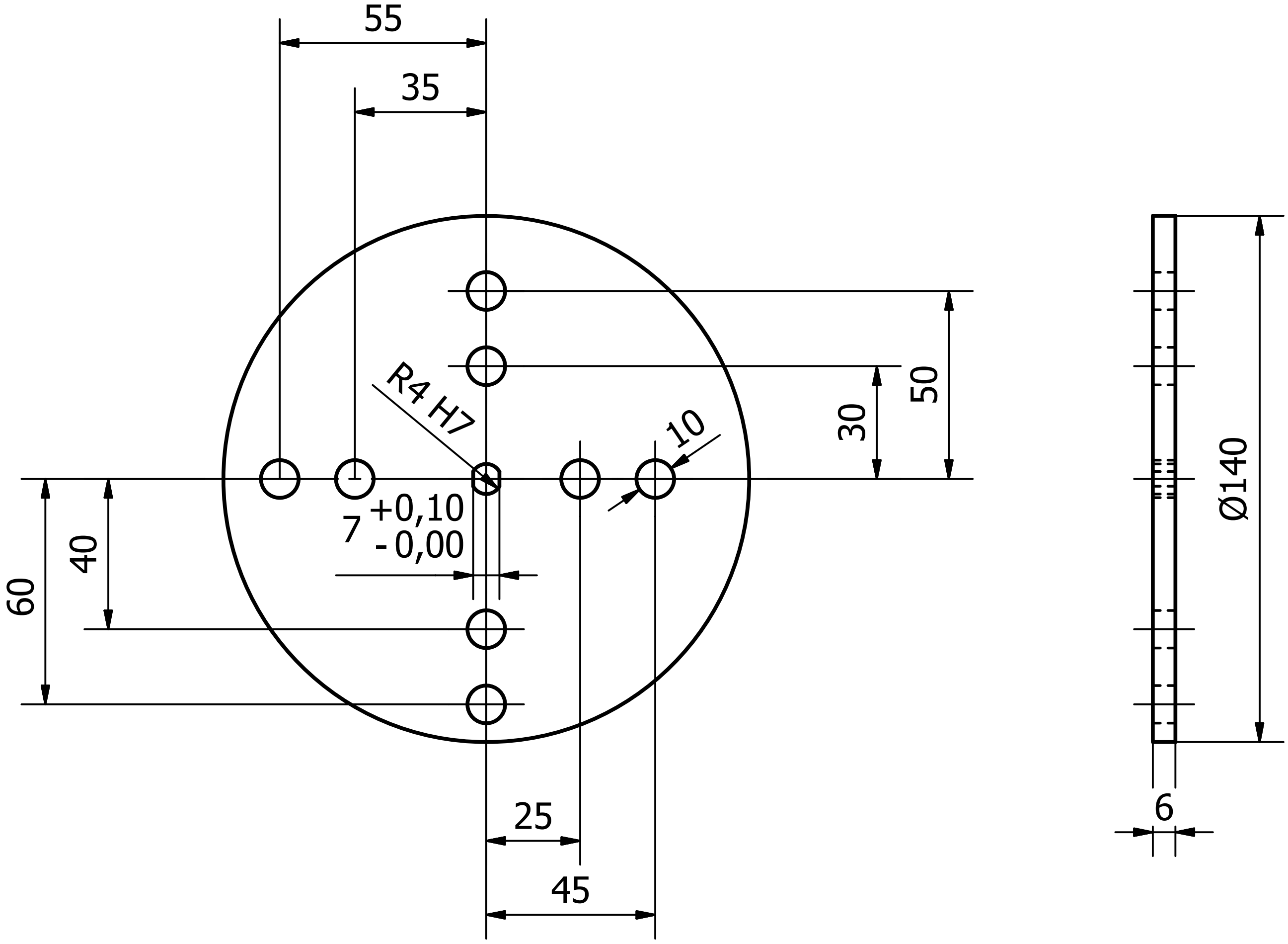
PARTS LIST

ITEM	QTY	PART NUMBER	DESCRIPTION	MATERIAL
1	1	Stepper Motor		Generic
2	1	Crank		Burch 6mm
3	1	Connecting Rod		Burch 6mm
4	1	Cylinder		Plastic
5	1	Guide		Burch 6mm
6	4	Support Axis		PLA
7	2	Box Plate		Burch 6mm
8	4	Box Piece 1		Burch 6mm
9	4	Box Piece 2		Burch 6mm
10	2	Box Piece 3		Burch 6mm
11	2	Bearings Axis		PLA
12	1	Transmisor		PLA
13	4	Vertical Threaded Axis		Steel
14	1	Fixing plate		Burch 6mm
15	1	Small Column		PLA
16	1	Big Column		PLA
17	1	Support Cylinder		PLA
18	4	Linear Bearing d=8mm		Steel
19	2	Rolling bearing B708 C GB/T 292-2007	Angular contact ball bearings d=8mm	Steel, Mild
20	2	ANSI B18.2.4.2M - M8x1.25	Metric Hex Nuts Styles 2	Steel, Mild
21	12	ANSI B18.2.4.2M - M4x0.6	Metric Hex Nuts Styles 2	Steel, Mild
22	8	ANSI B18.2.4.2M - M5x0.8	Metric Hex Nuts Styles 2	Steel, Mild
23	12	IFI 502 - M4x0.6 x 25	Hex Head Tapping Screw - Metric	Steel, Mild
24	2	ISO 4014 - M8 x 70	Hex-Head Bolt	Stainless Steel, 440C
25	4	AS 1110 - M5 x 16	ISO metric hexagon precision screws	Steel, Mild
26	1	Washer d=8mm		Generic
27	1	Washer d=10mm		Generic
28	20	Washer d=4mm		Generic
29	4	AS 1110 - M2 x 6	ISO metric hexagon precision screws	Steel, Mild

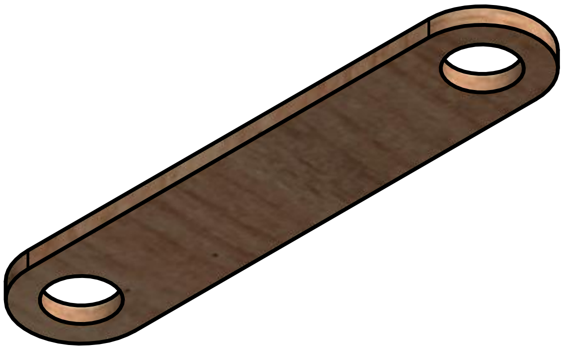
Designed by	Checked by	Approved by	Date	Date	Scale: 1:3
mhermoso				2/05/2016	

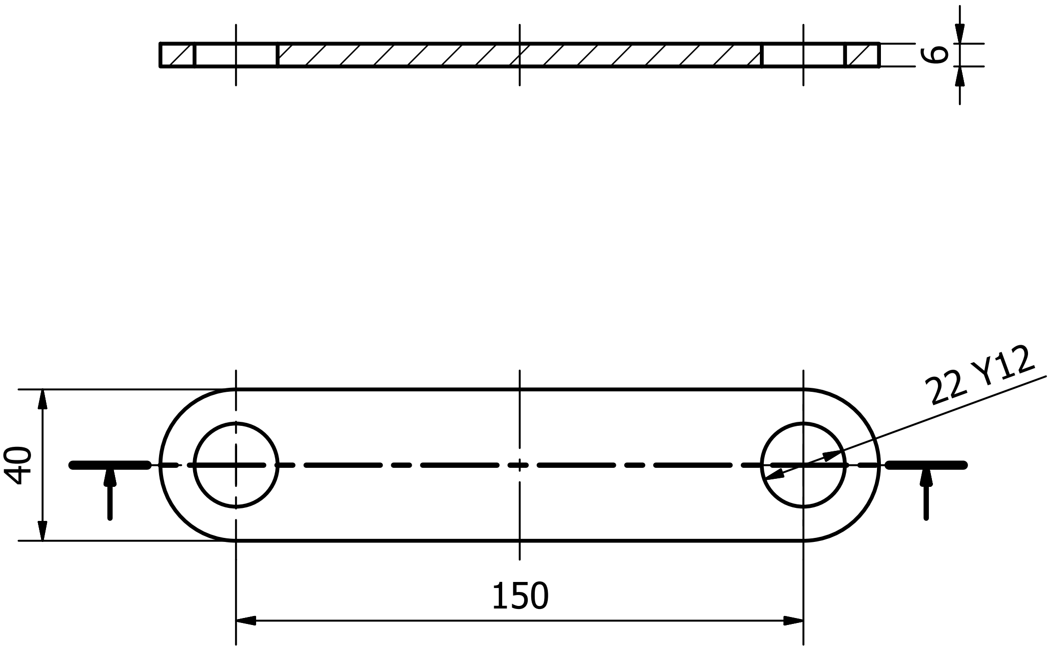
S3. Explotion	Miguel Hermoso de Mendoza Sarasa		
	S3. Driving Mechanism	Edition	Sheet 2 / 17



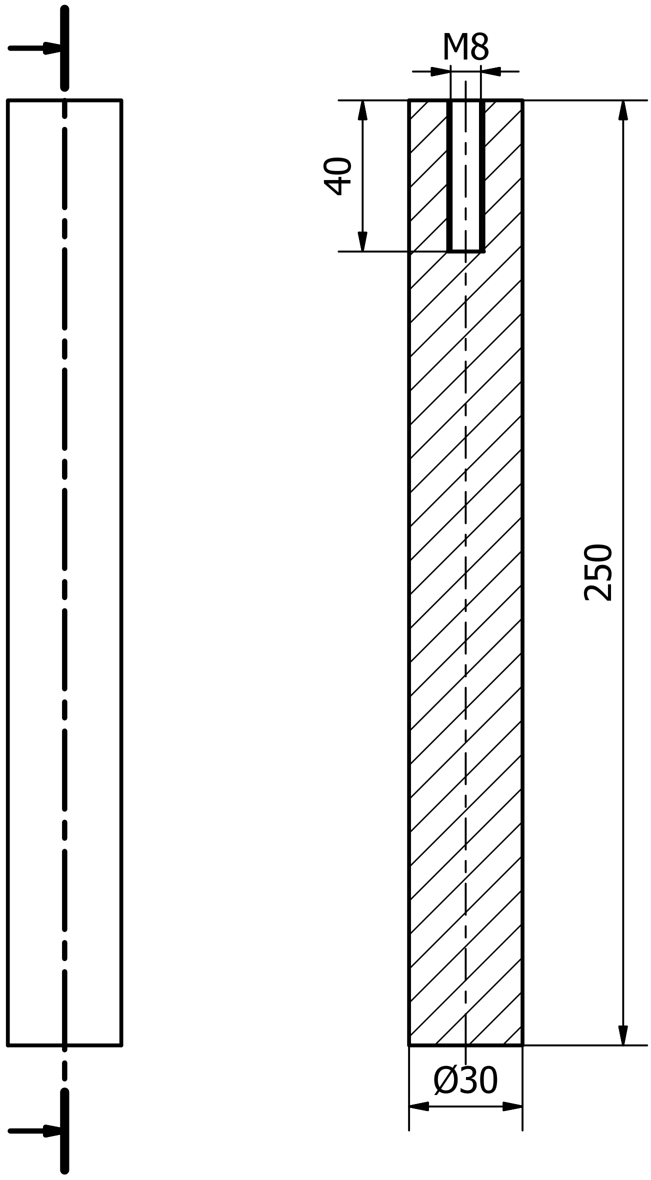


Designed by	Checked by	Approved by	Date		Date	Scale: 1:2
mhermoso					2/05/2016	
S3.2. Crank			Miguel Hermoso de Mendoza Sarasa			
			S3. Driving Mechanism		Edition	Sheet 3 / 17



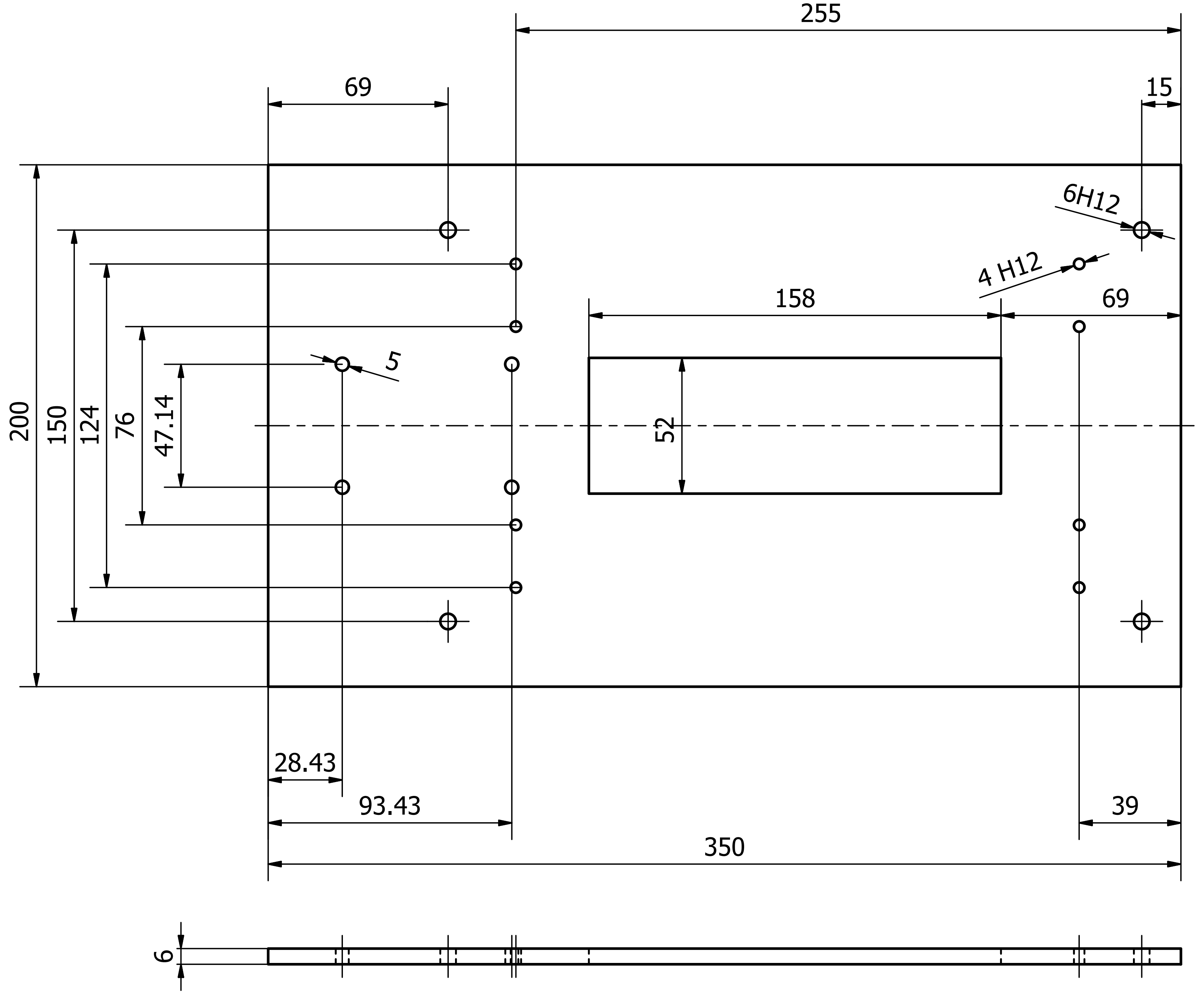


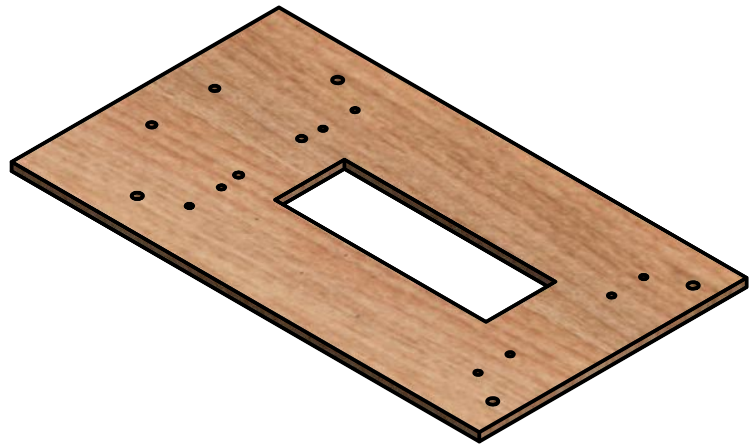
Designed by	Checked by	Approved by	Date		Date	Scale: 1:1
mhermoso					2/05/2016	
S3.3. Connecting Rod			Miguel Hermoso de Mendoza Sarasa			
			S3. Driving Mechanism		Edition	Sheet 4 / 17



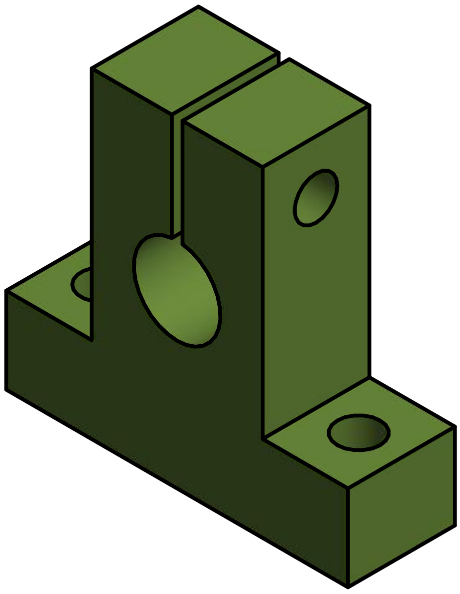
Designed by mhermoso	Checked by	Approved by	Date	Date 2/05/2016	Scale: 1:2

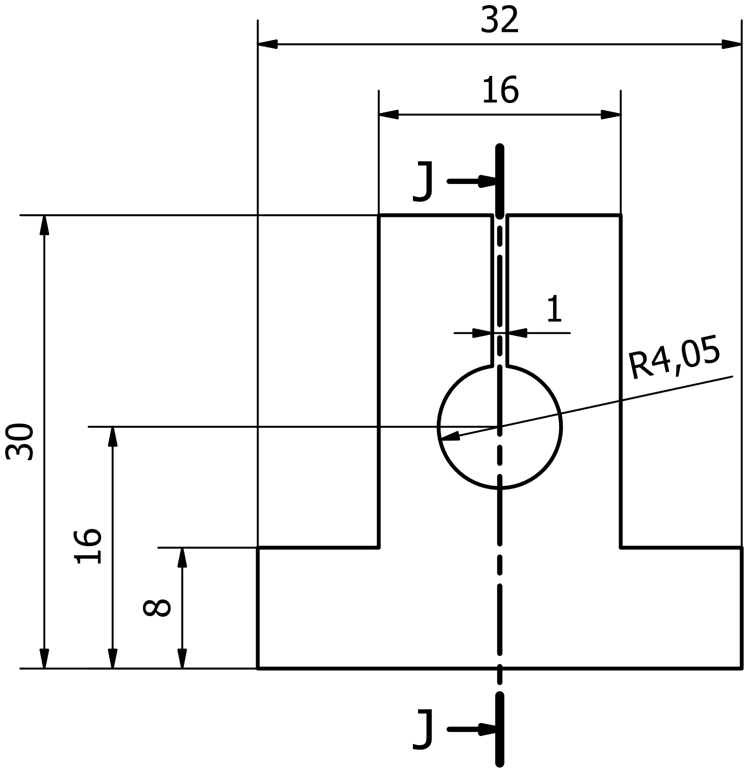
S3.4. Cylinder	Miguel Hermoso de Mendoza Sarasa		
	S3. Driving Mechanism	Edition	Sheet 5 / 17

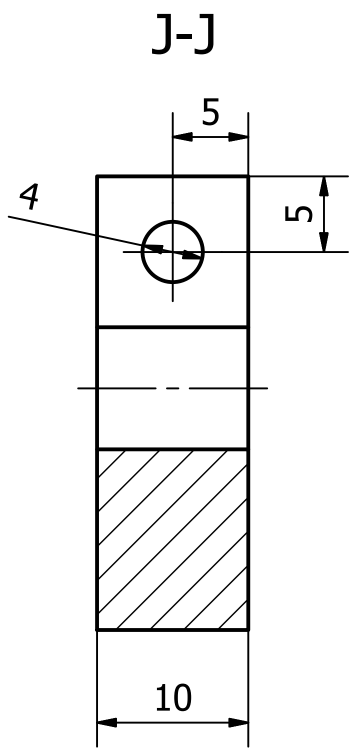




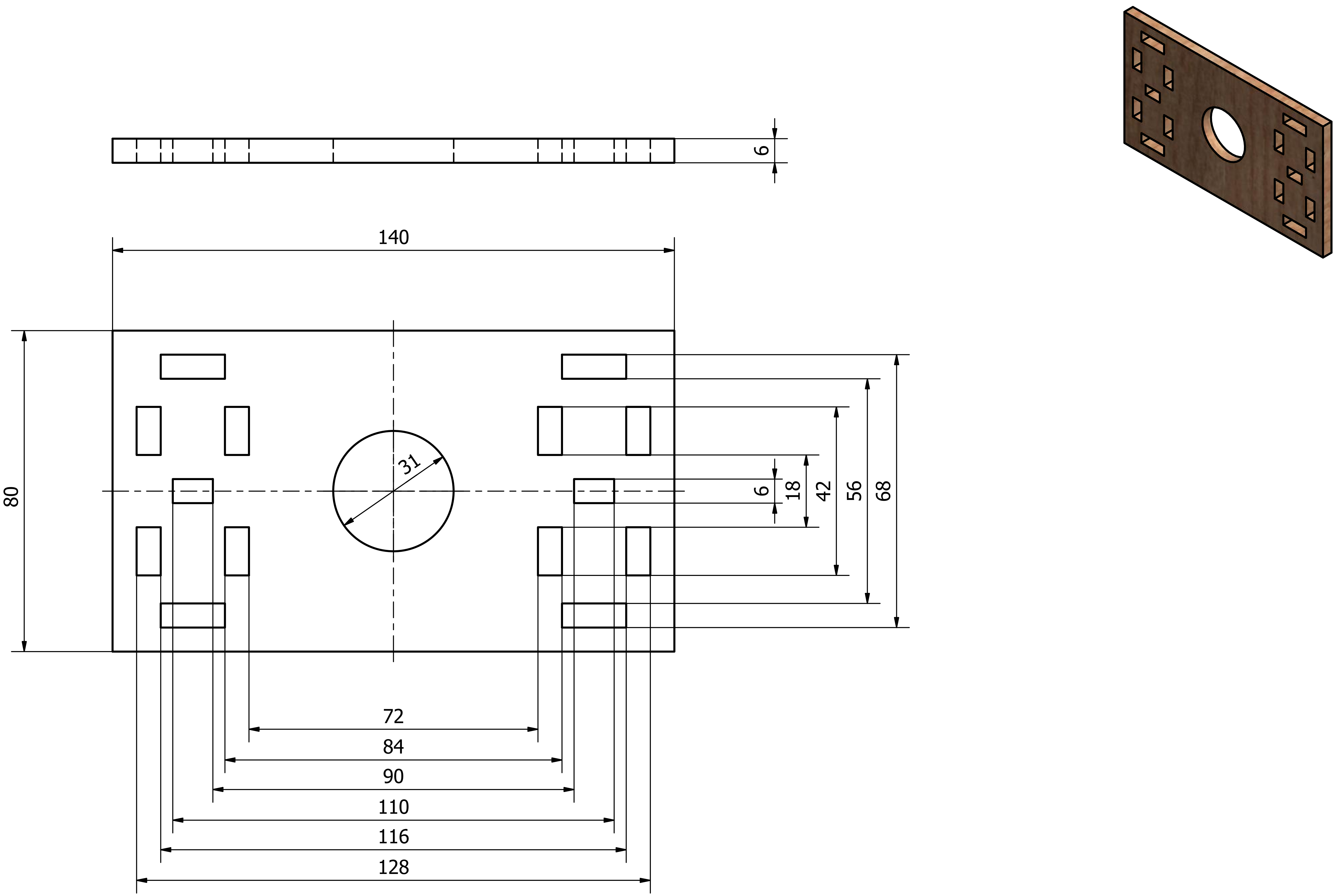
Designed by	Checked by	Approved by	Date		Date	Scale: 1:2
mhermoso					2/05/2016	
S3.5. Guide			Miguel Hermoso de Mendoza Sarasa			
			S3. Driving Mechanism		Edition	Sheet 6 / 17



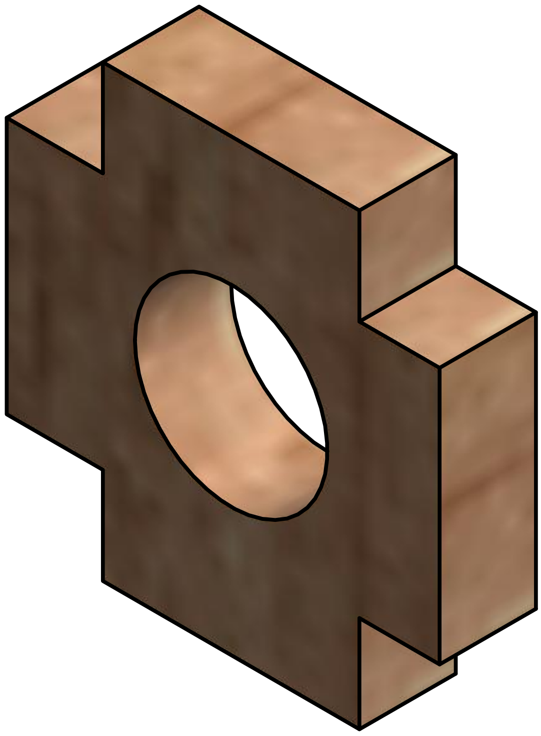


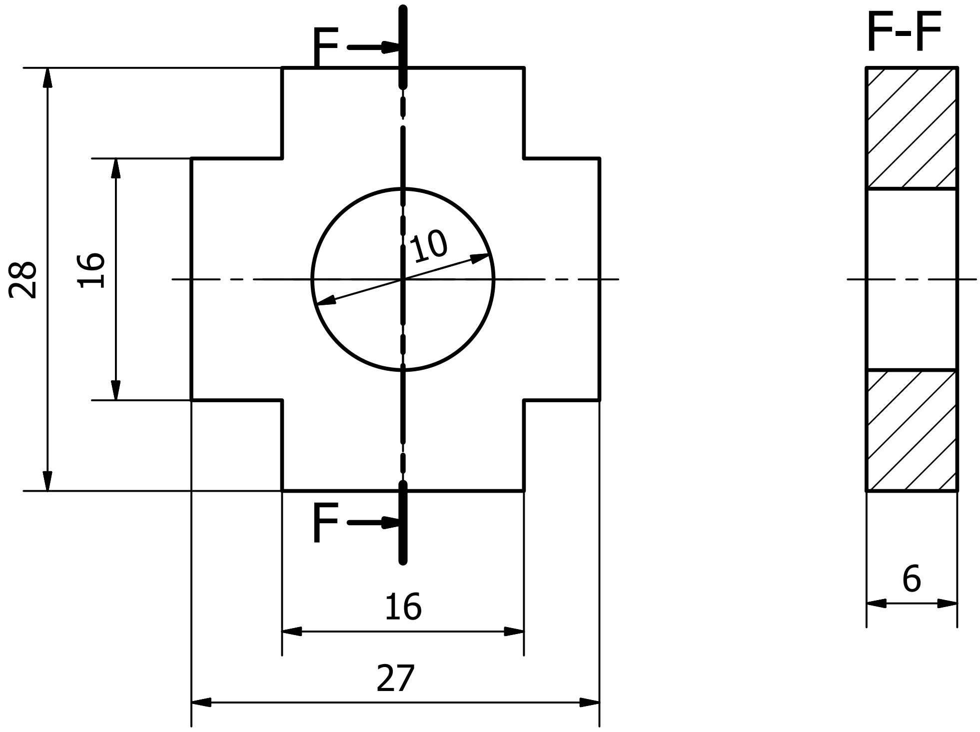


Designed by mhermoso	Checked by	Approved by	Date	Date 2/05/2016	Scale: 2:1
S3.6. Support Axis		Miguel Hermoso de Mendoza Sarasa			
		S3. Driving Mechanism		Edition	Sheet 7 / 17

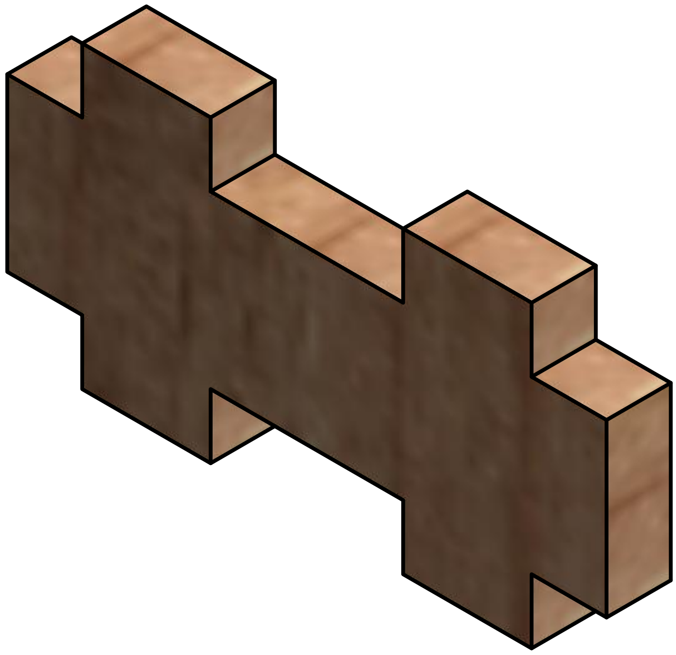


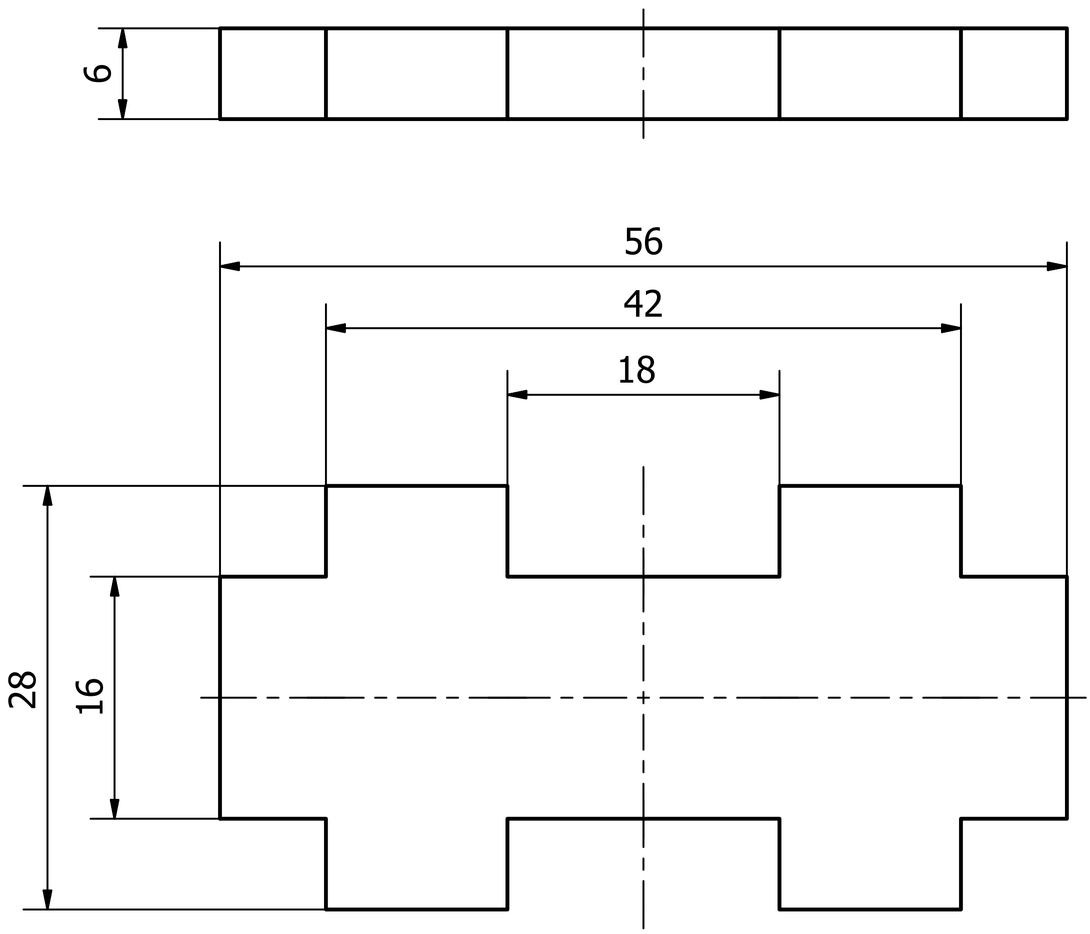
Designed by	Checked by	Approved by	Date	Date	Scale: 1:1
mhermoso				2/05/2016	
S3.7. Box Plate	Miguel Hermoso de Mendoza Sarasa				
	S3. Driving Mechanism			Edition	Sheet 8 / 17



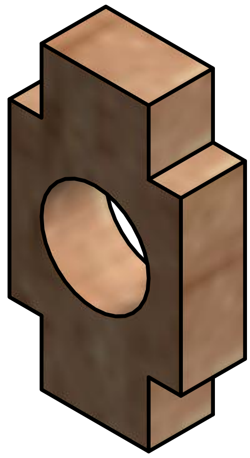


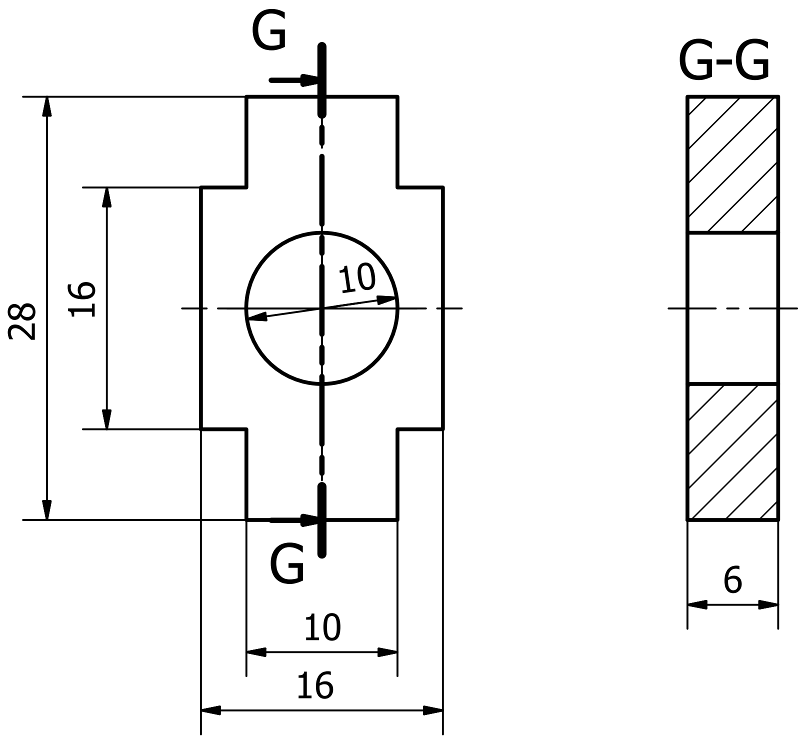
Designed by	Checked by	Approved by	Date		Date	Scale: 2:1
mhermoso					2/05/2016	
S3.8. Box Piece 1			Miguel Hermoso de Mendoza Sarasa			
			S3. Driving Mechanism		Edition	Sheet 9 / 17



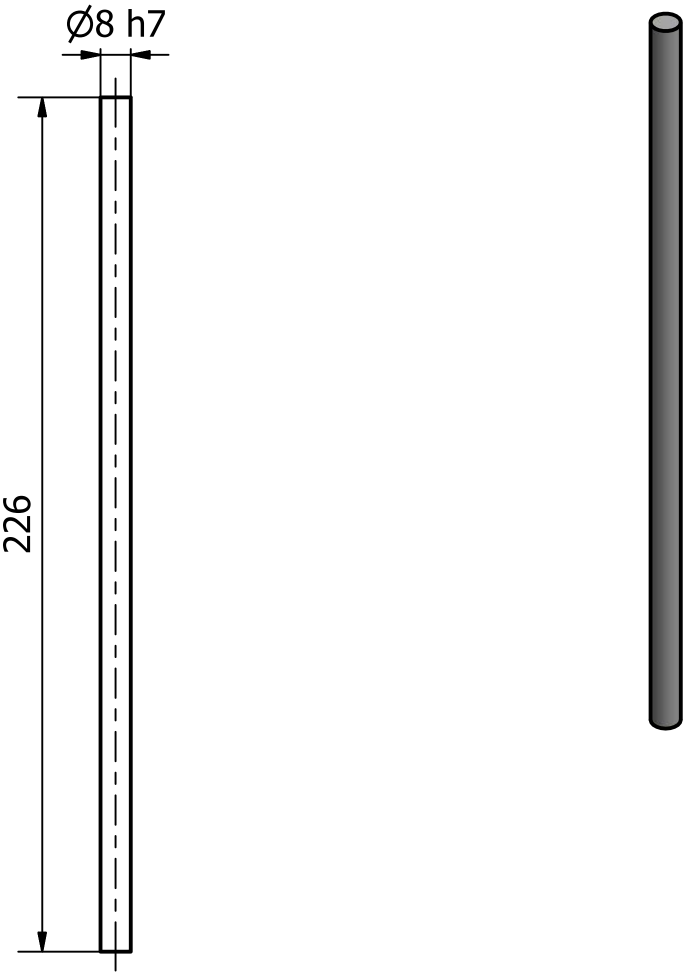


Designed by mhermoso	Checked by	Approved by	Date	Date 2/05/2016	Scale: 2:1
S3.9. Box Piece 2	Miguel Hermoso de Mendoza Sarasa				
	S3. Driving Mechanism			Edition	Sheet 10 / 17



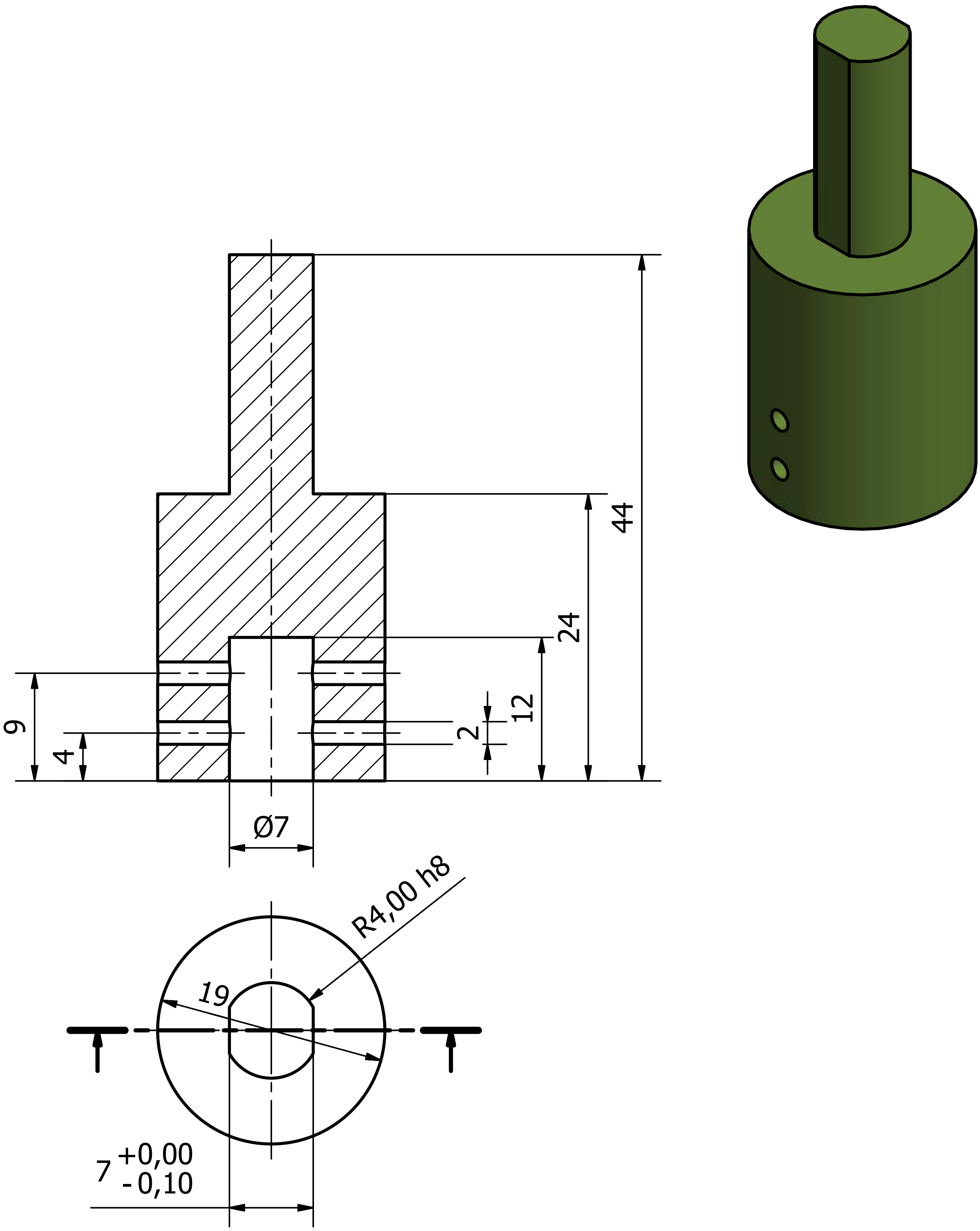


Designed by mhermoso	Checked by	Approved by	Date	Date 2/05/2016	Scale: 2:1
S3.10. Box Piece 3		Miguel Hermoso de Mendoza Sarasa			
		S3. Driving Mechanism		Edition	Sheet 11 / 17

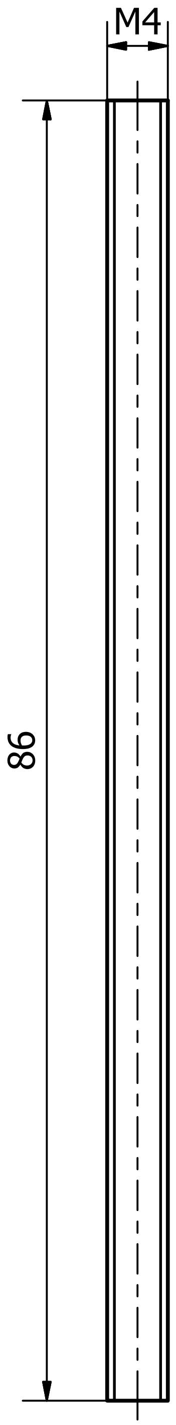


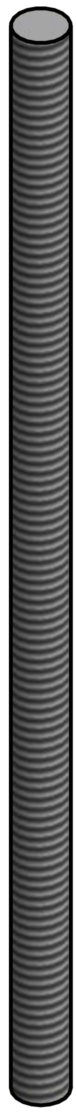
Designed by	Checked by	Approved by	Date		Date	Scale: 1:2
mhermoso					2/05/2016	

S3.11. Bearings Axis	Miguel Hermoso de Mendoza Sarasa		
	S3. Driving Mechanism	Edition	Sheet 12 / 17

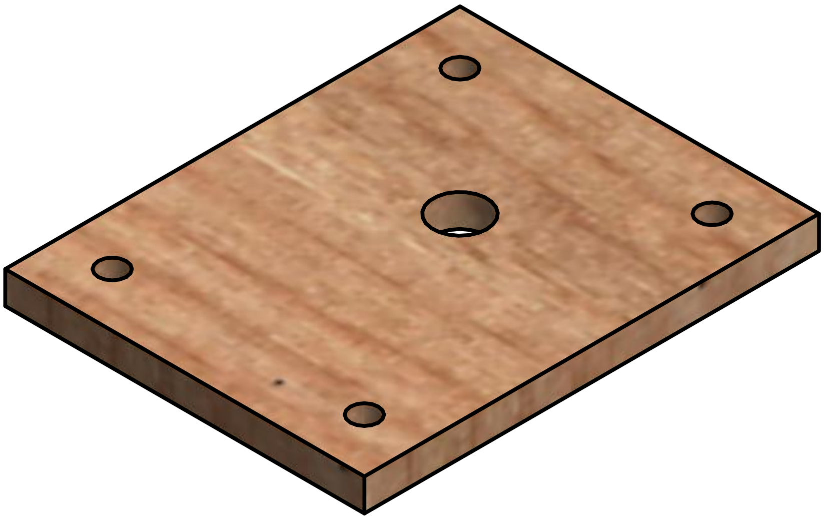


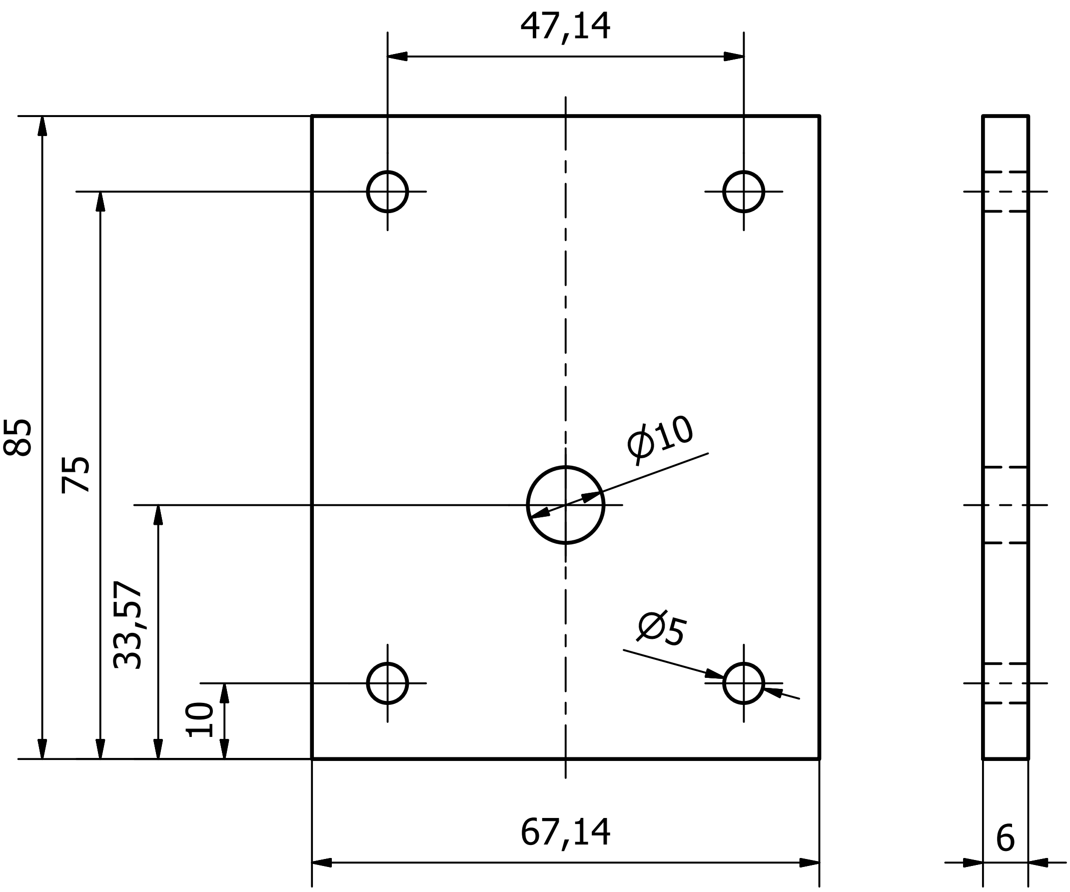
Designed by mhermoso	Checked by	Approved by	Date		Date 2/05/2016	Scale: 2:1
S3.12. Transmisor			Miguel Hermoso de Mendoza Sarasa			
			S3. Driving Mechanism		Edition	Sheet 13 / 17



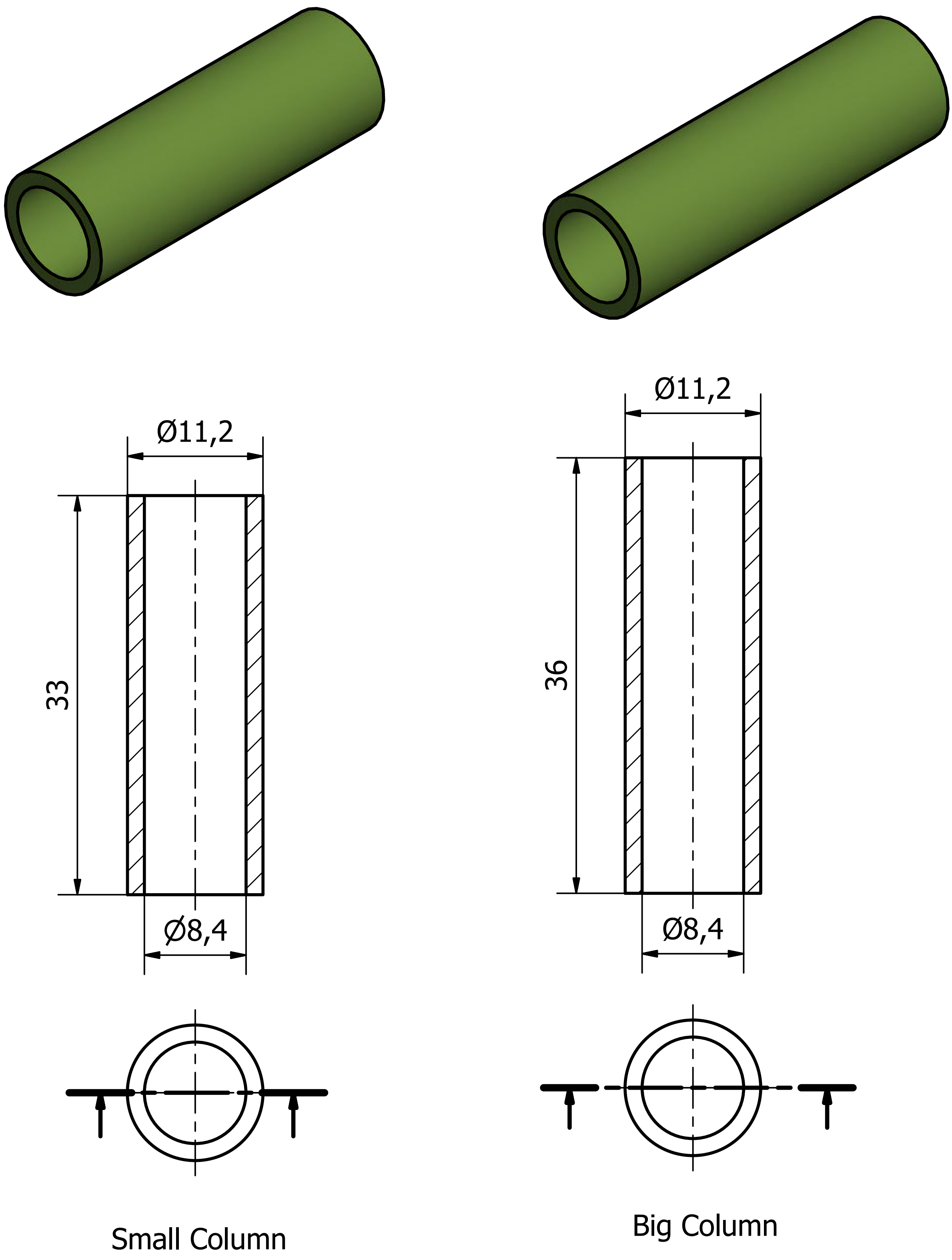


Designed by mhermoso	Checked by	Approved by	Date	Date 2/05/2016	Scale: 2:1
S3.13. Vertical Threaded Axis	Miguel Hermoso de Mendoza Sarasa				
	S3. Driving Mechanism			Edition	Sheet 14 / 17

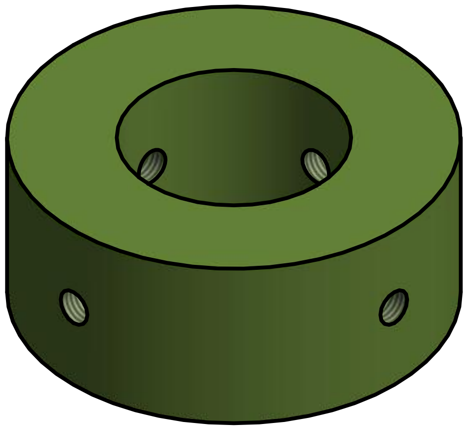


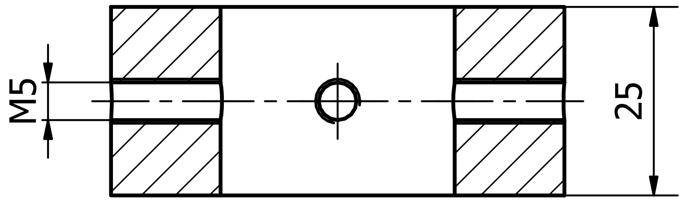


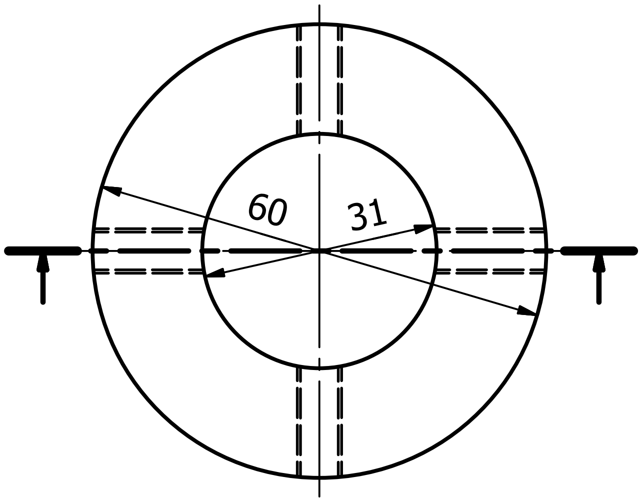
Designed by mhermoso	Checked by	Approved by	Date	Date 2/05/2016	Scale: 1:1
S3.14. Fixing Plate		Miguel Hermoso de Mendoza Sarasa S3. Driving Mechanism		Edition	Sheet 15 / 17



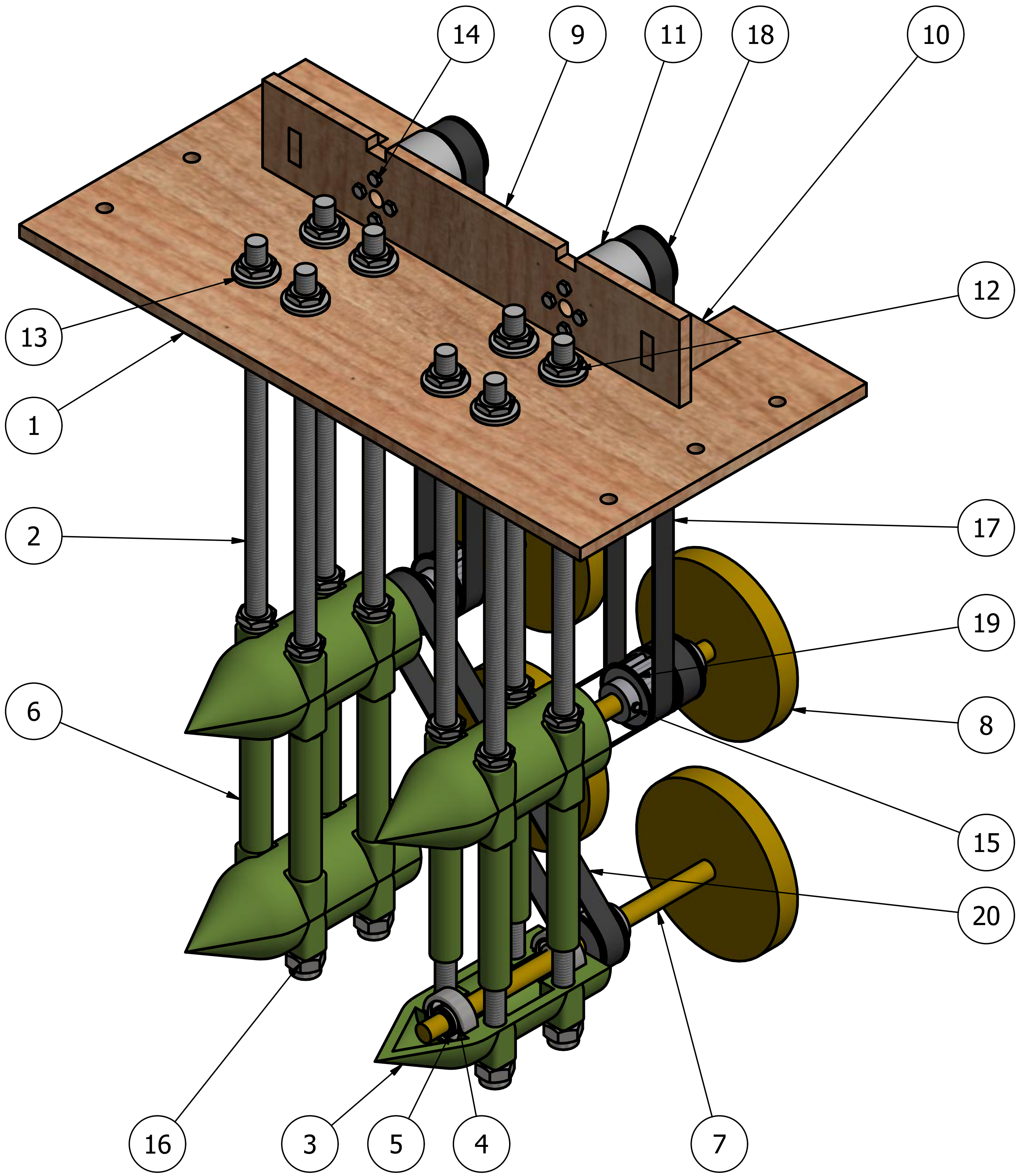
Designed by mhermoso	Checked by	Approved by	Date		Date 2/05/2016	Scale: 2:1
S3.15. Small Column and S3.16. Big Column			Miguel Hermoso de Mendoza Sarasa			
			S3. Driving Mechanism		Edition	Sheet 16 / 17







Designed by	Checked by	Approved by	Date		Date	Scale: 1:1
mhermoso					2/05/2016	
S3.17. Support Cylinder	Miguel Hermoso de Mendoza Sarasa					
	S3. Driving Mechanism				Edition	Sheet 17 / 17



PARTS LIST

ITEM	QTY	PART NUMBER	DESCRIPTION	MATERIAL
1	1	Basis		Burch 6mm
2	8	Vertical Threaded Bar		Stainless steel
3	8	Bullet		PLA
4	8	DIN 615 - E 8 - 8 x 22 x 8	Magneto Ball Bearing d=8mm	Steel, Mild
5	8	DIN 471 - 8x0.8	Retaining rings for shaft	Steel, Mild
6	8	Column		PLA
7	4	Axis Propellers		Brass
8	4	Propeller		Brass
9	1	Plate Supporting Motors		Burch
10	2	Nerv		Burch
11	2	Brushless Motor		
12	24	DIN 439 - M8	Hex Nut	Steel, Mild
13	24	Washer d=8mm		Steel
14	8	AS 1110 - M3 x 8	ISO metric hexagon screws	Steel, Mild
15	12	AS 1110 - M2 x 6	ISO metric hexagon screws	Steel, Mild
16	8	ISO 10512 - M8 x 1	Hex Nut	Steel, Mild
17	2	Timing Belt HTD, Pitch=5mm	90 Tooth, 450mm Length x 9mm Width	Rubber
18	2	Motor Pulley	11.5 T5 x 16	PLA
19	6	Axis Pulley	11.5 T5 x 16	PLA
20	2	Metric Pitch Timing belt T5 (Pitch=5mm)	73 Tooth, 365mm Length X 10mm Width	Rubber

Designed by	Checked by	Approved by	Date	Date	Scale: 1:2
mhermoso				18/05/2016	

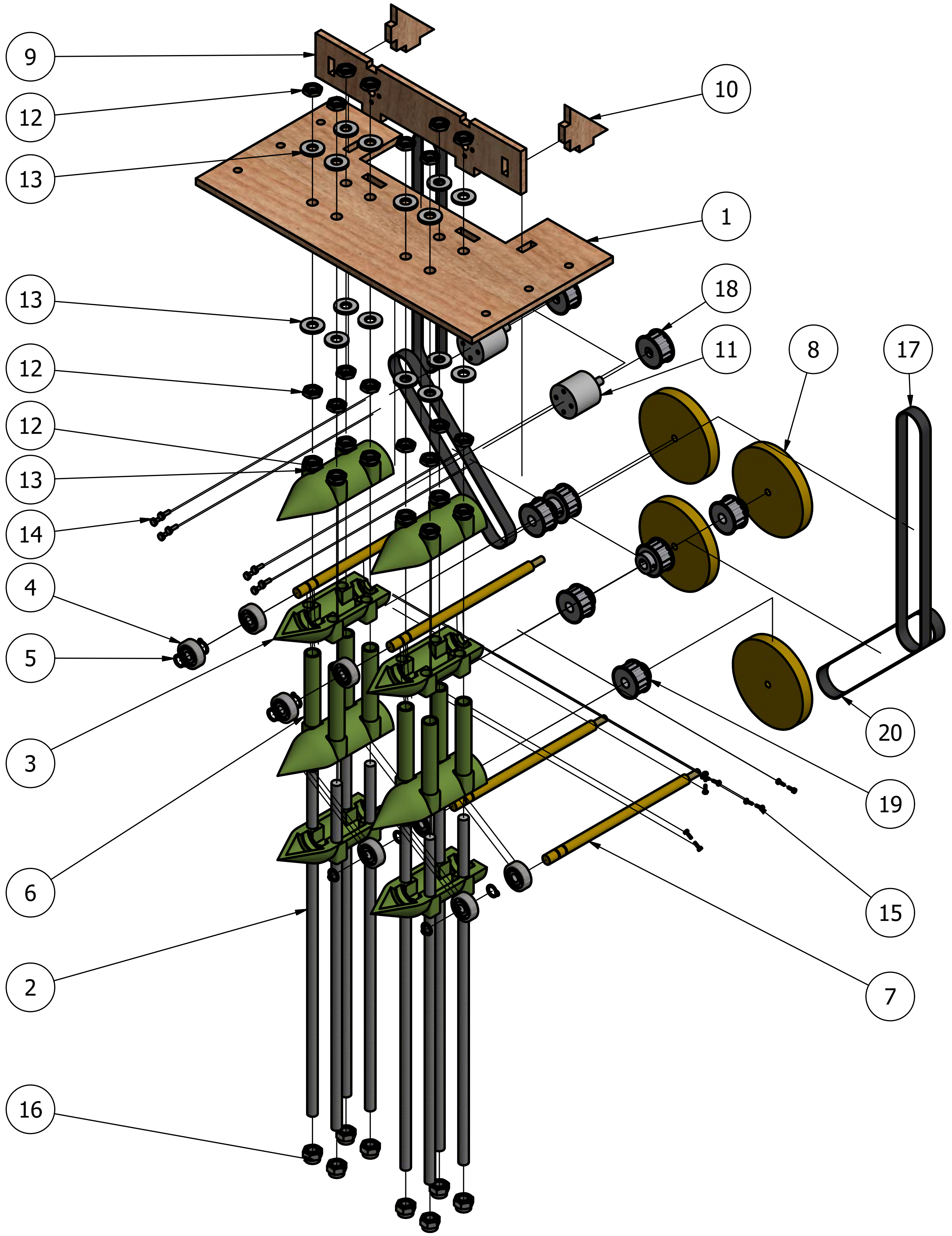
V.I.V experiment set up

Miguel Hermoso de Mendoza Sarasa

S4. Propellers Structure

Edition

Sheet 1 / 12



PARTS LIST

ITEM	QTY	PART NUMBER	DESCRIPTION	MATERIAL
1	1	Basis		Burch 6mm
2	8	Vertical Threaded Bar		Stainless steel
3	8	Bullet		PLA
4	8	DIN 615 - E 8 - 8 x 22 x 8	Magneto Ball Bearing d=8mm	Steel, Mild
5	8	DIN 471 - 8x0.8	Retaining rings for shaft	Steel, Mild
6	8	Column		PLA
7	4	Axis Propellers		Brass
8	4	Propeller		Brass
9	1	Plate Supporting Motors		Burch
10	2	Nerv		Burch
11	2	Brushless Motor		
12	24	DIN 439 - M8	Hex Nut	Steel, Mild
13	24	Washer d=8mm		Steel
14	8	AS 1110 - M3 x 8	ISO metric hexagon screws	Steel, Mild
15	12	AS 1110 - M2 x 6	ISO metric hexagon screws	Steel, Mild
16	8	ISO 10512 - M8 x 1	Hex Nut	Steel, Mild
17	2	Timing Belt HTD, Pitch=5mm	90 Tooth, 450mm Length x 9mm Width	Rubber
18	2	Motor Pulley	11.5 T5 x 16	PLA
19	6	Axis Pulley	11.5 T5 x 16	PLA
20	2	Metric Pitch Timing belt T5 (Pitch=5mm)	73 Tooth, 365mm Length X 10mm Width	Rubber

Designed by	Checked by	Approved by	Date	Date
mhermoso				18/05/2016

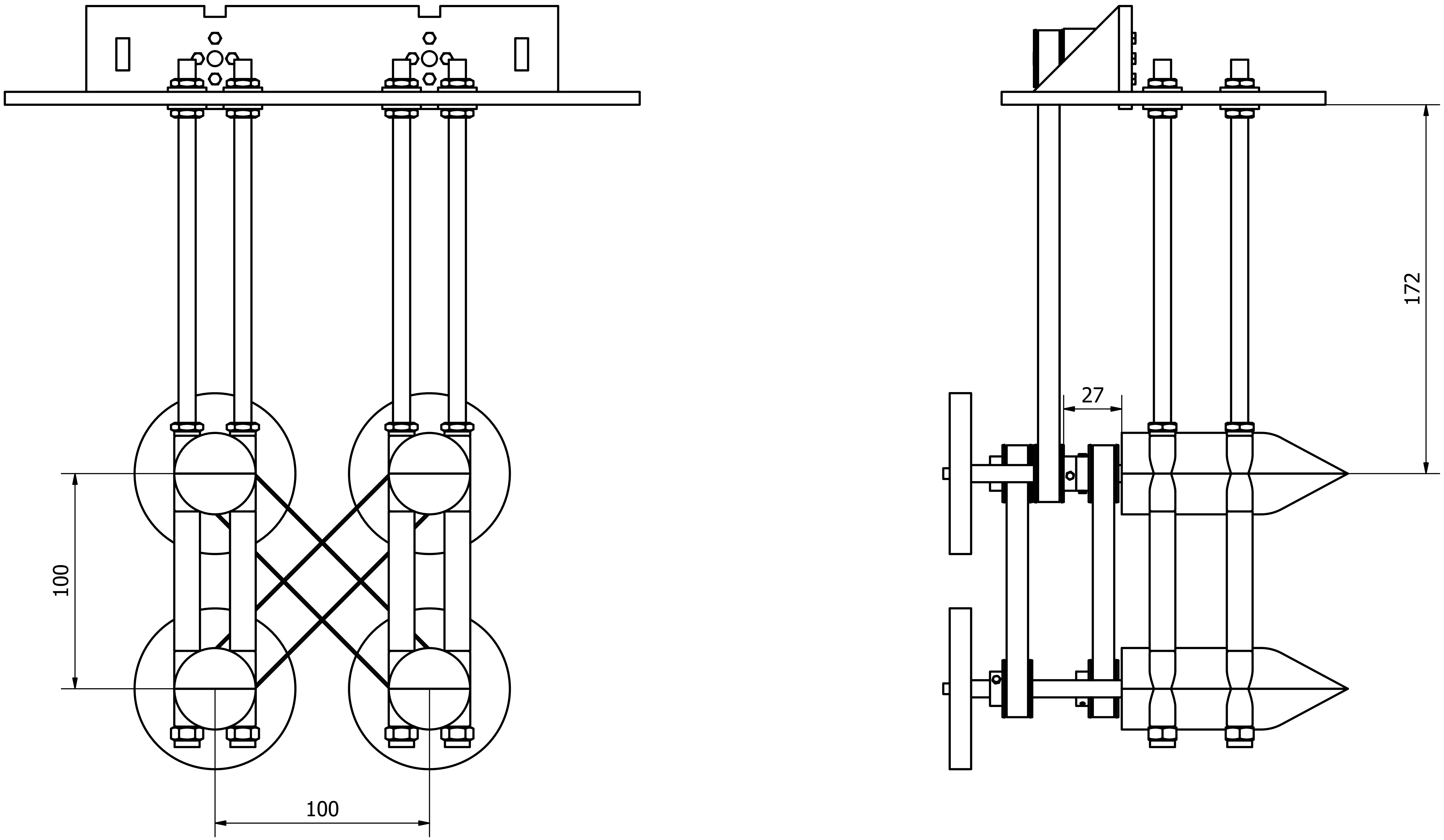
S4. Explosion

Miguel Hermoso de Mendoza Sarasa

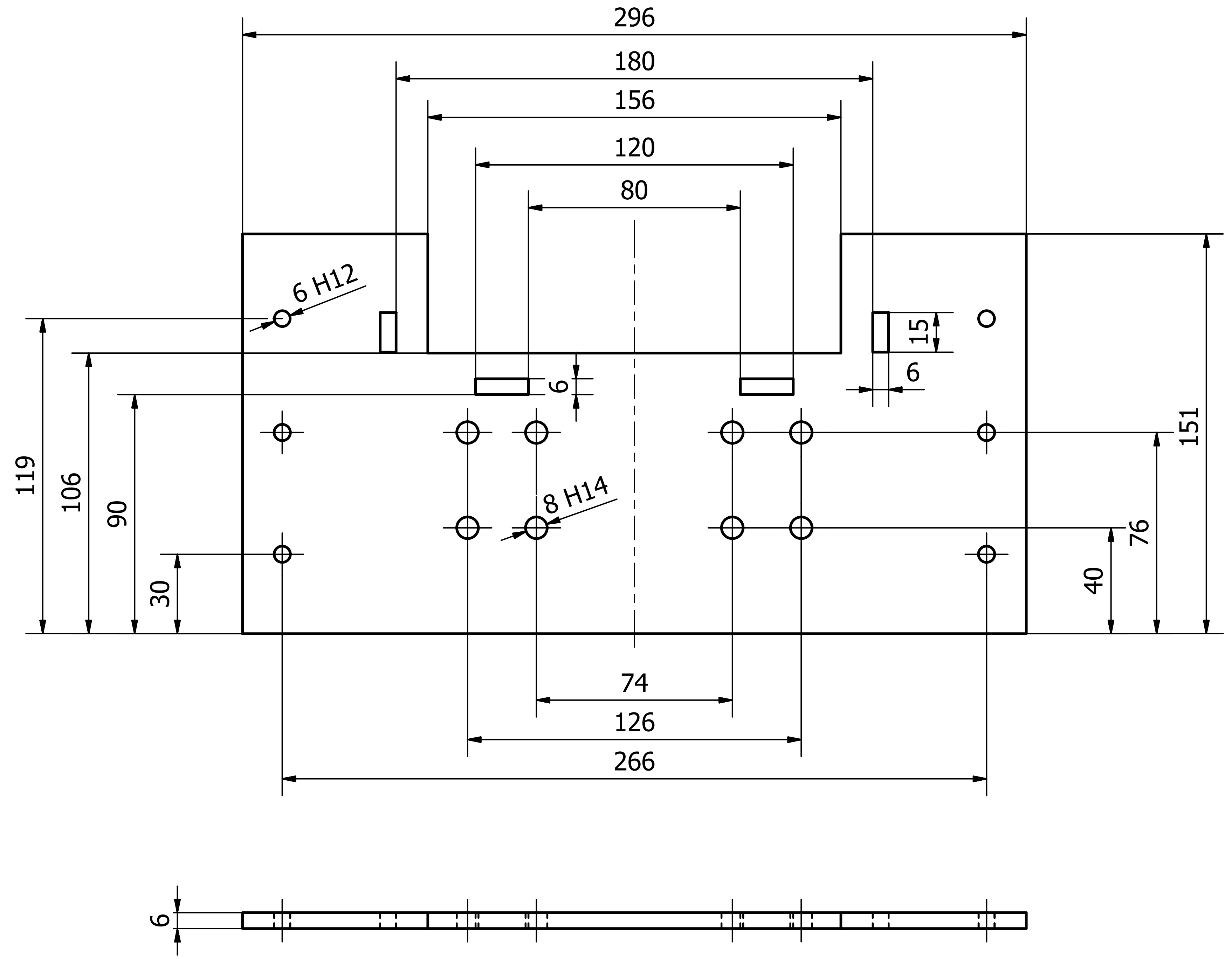
S4. Propellers Structure

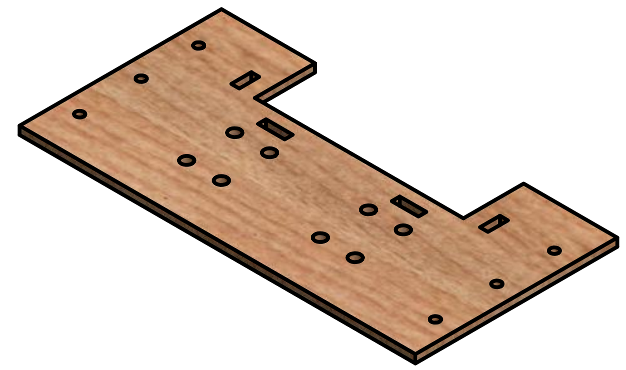
Edition

Sheet 2 / 12

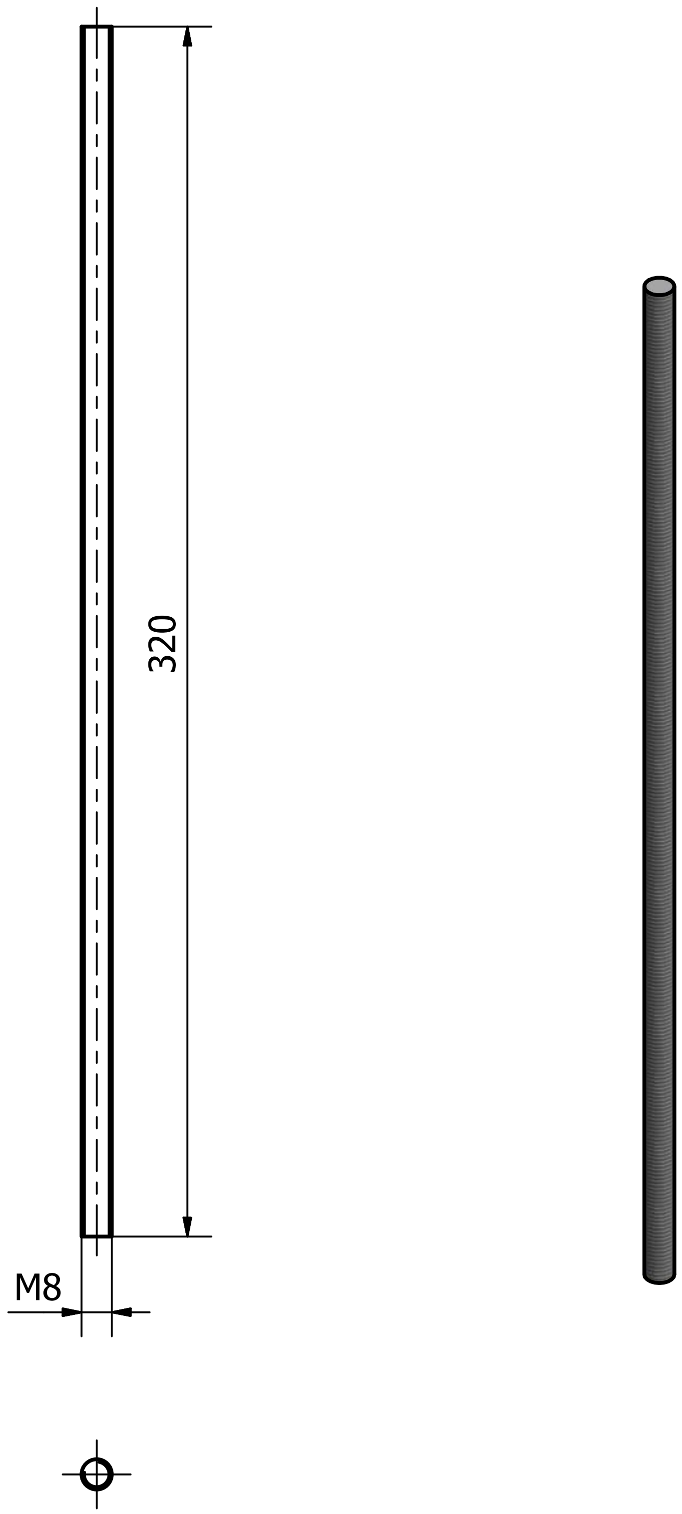


Designed by mhermoso	Checked by	Approved by Date	Date 18/05/2016	Scale: 1:2
S4. Views		Miguel Hermoso de Mendoza Sarasa		
		S4. Propellers Structure	Edition	Sheet 3 / 12



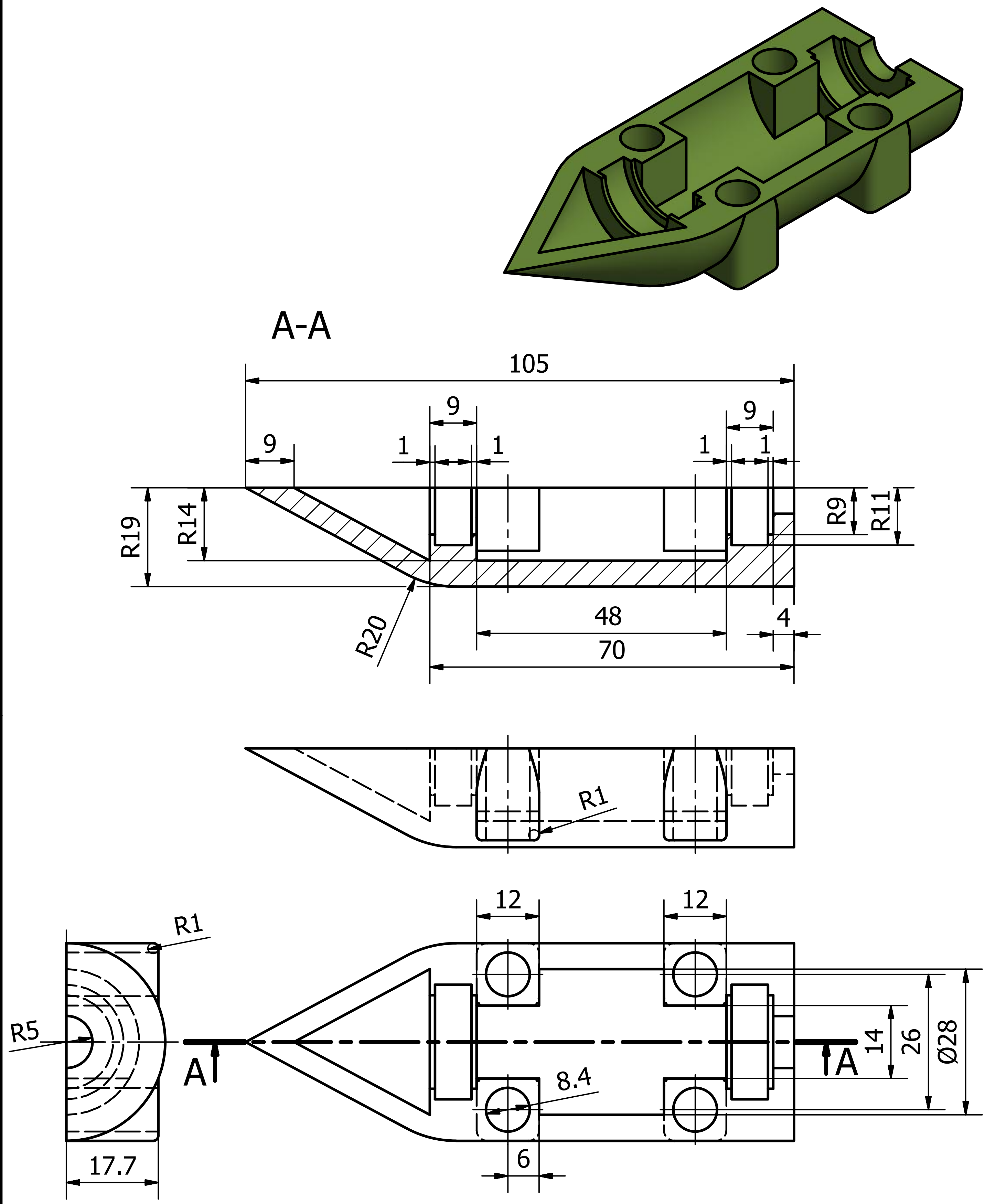


Designed by mhermoso	Checked by	Approved by	Date	Date 18/05/2016	Scale: 1:2
S4.1. Basis		Miguel Hermoso de Mendoza Sarasa			
		S4. Propellers Structure		Edition	Sheet 4 / 12

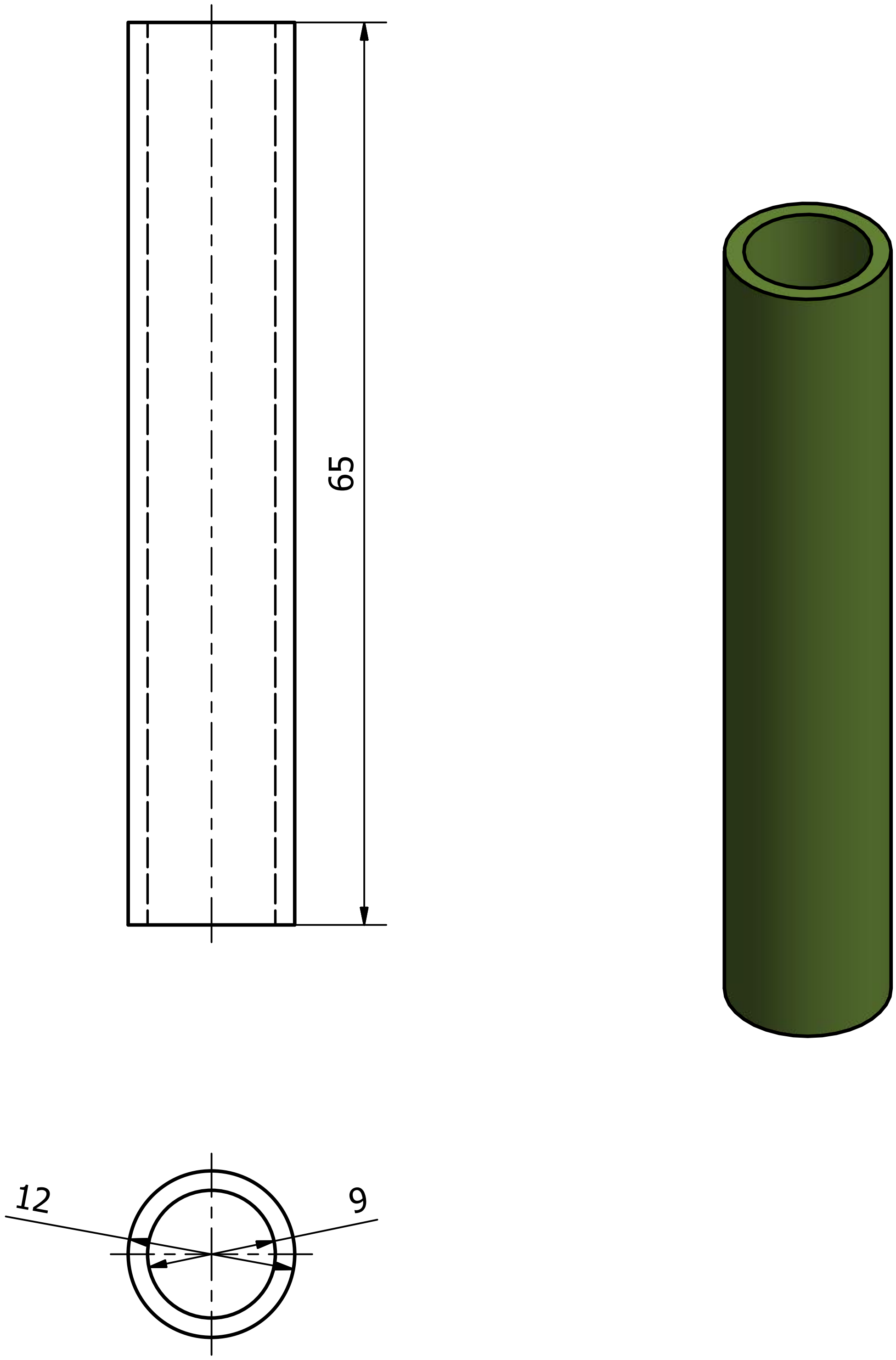


Designed by	Checked by	Approved by	Date		Date	Scale: 1:2
mhermoso					18/05/2016	

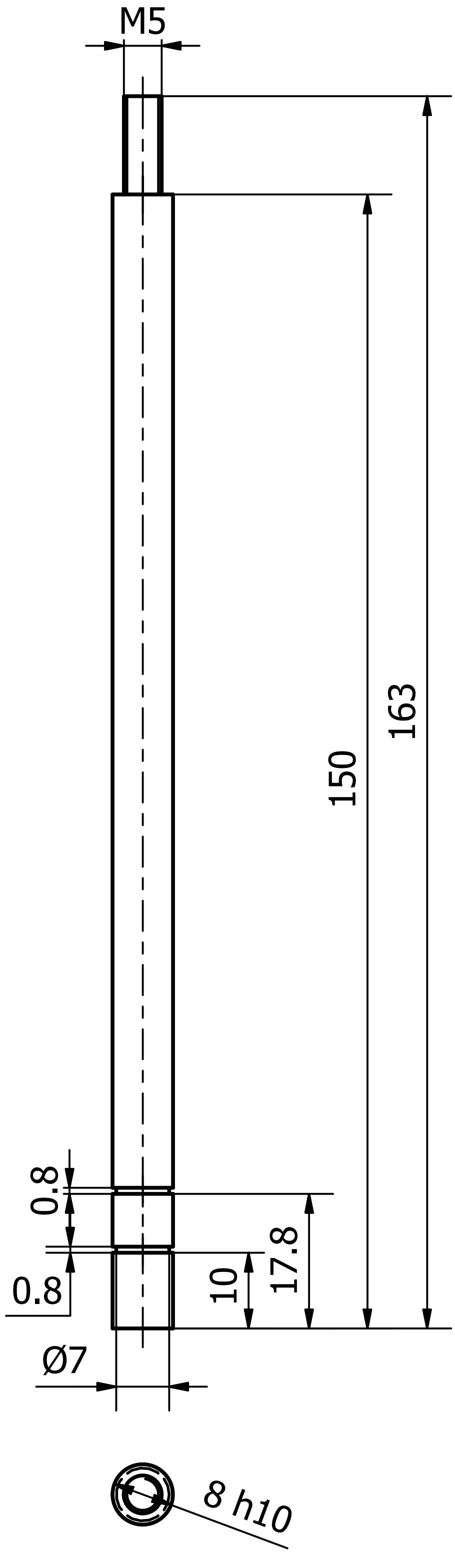
S4.2. Vertical Threaded Bar	Miguel Hermoso de Mendoza Sarasa		
	S4. Propellers Structure	Edition	Sheet 5 / 12

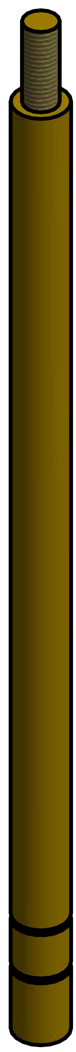


Designed by	Checked by	Approved by	Date		Date	Scale: 1:1
mhermoso					18/05/2016	
S4.3. Bullet			Miguel Hermoso de Mendoza Sarasa			
			S4. Propellers Structure		Edition	Sheet 6 / 12

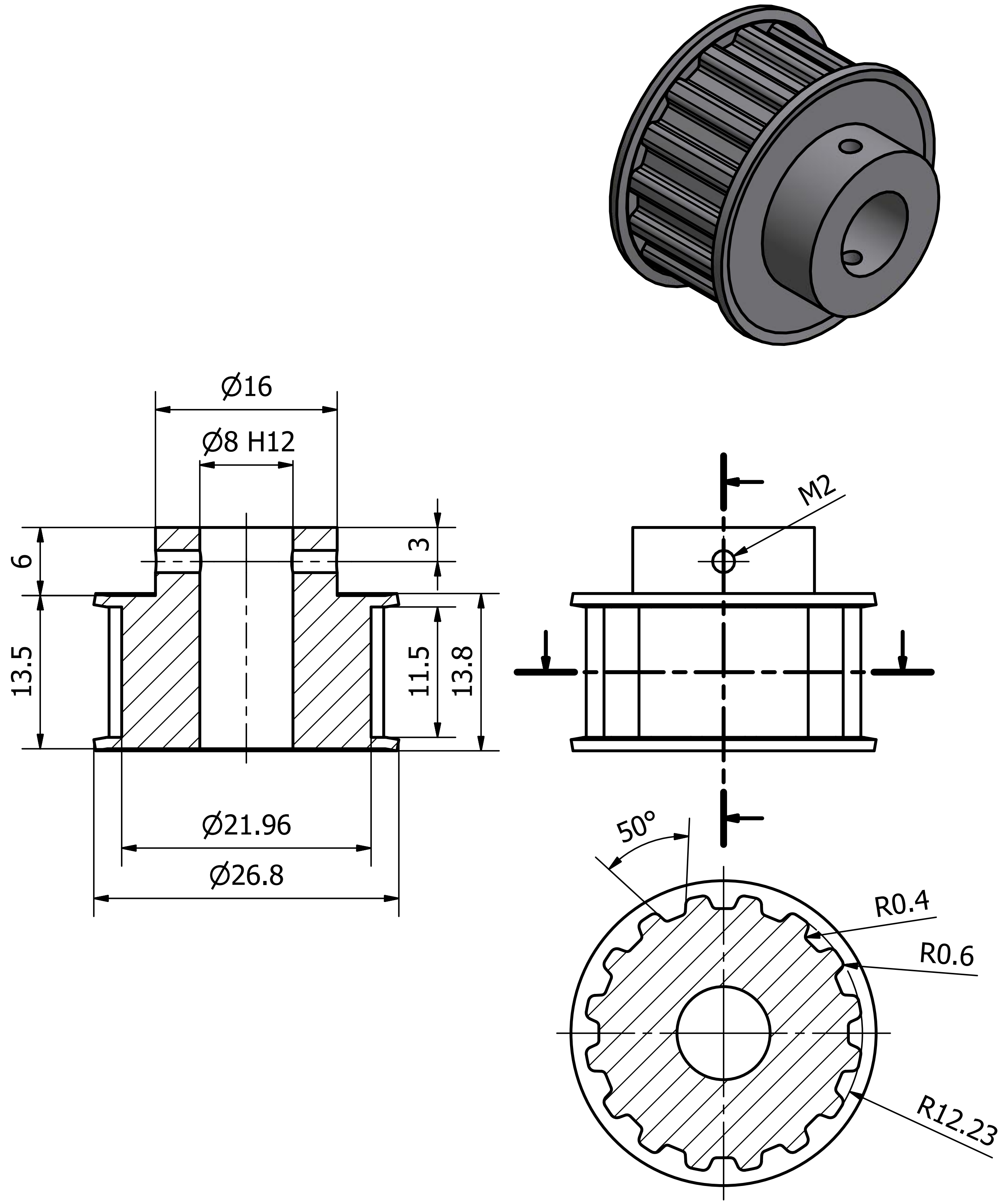


Designed by mhermoso	Checked by	Approved by	Date	Date 18/05/2016	Scale: 2:1
S4.6. Column	Miguel Hermoso de Mendoza Sarasa				
	S4. Propellers Structure			Edition	Sheet 7 / 12



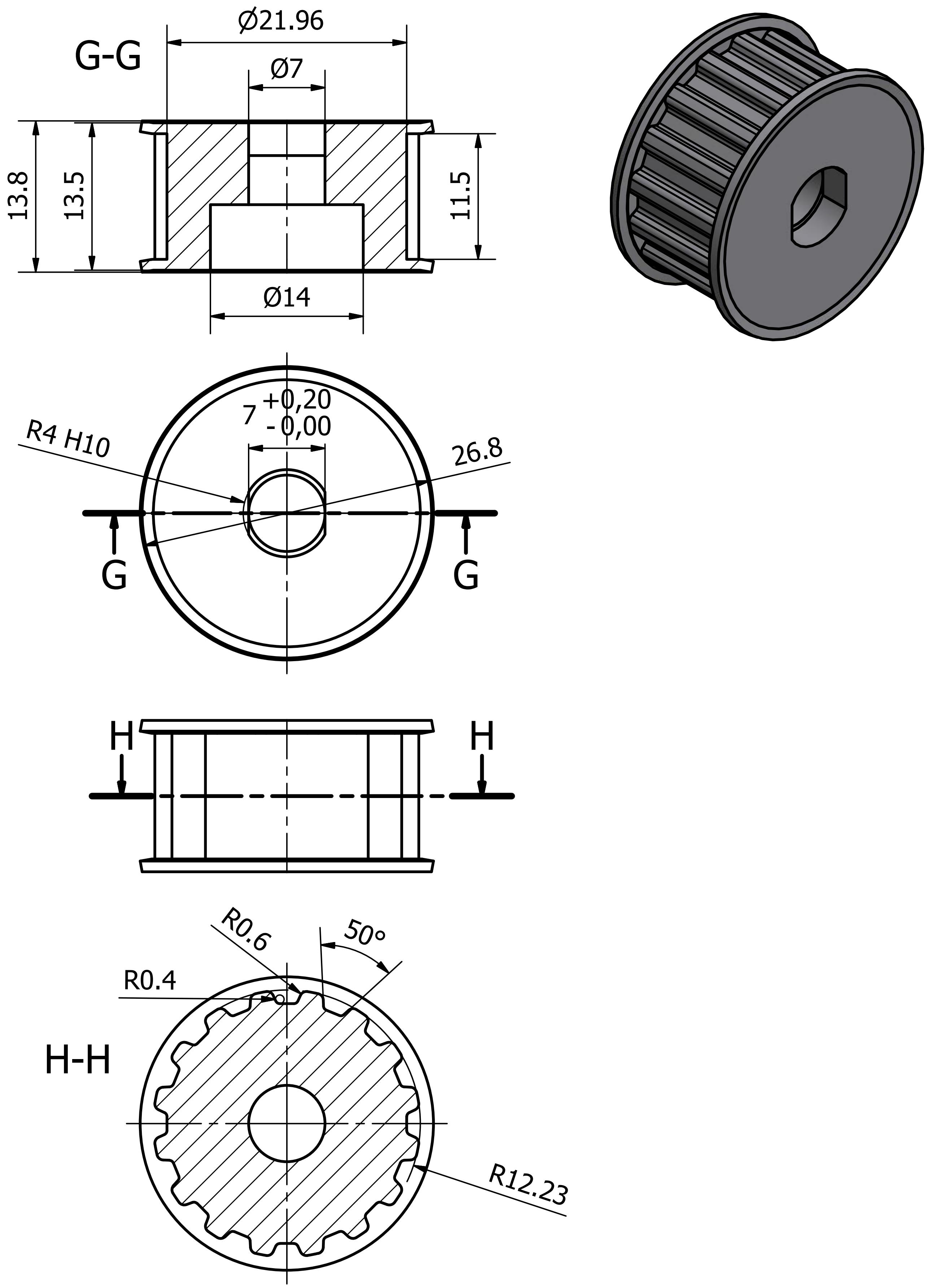


Designed by	Checked by	Approved by	Date		Date	Scale: 1:1
mhermoso					18/05/2016	
S4.7. Axis Propellers			Miguel Hermoso de Mendoza Sarasa			
			S4. Propellers Structure		Edition	Sheet 8 / 12

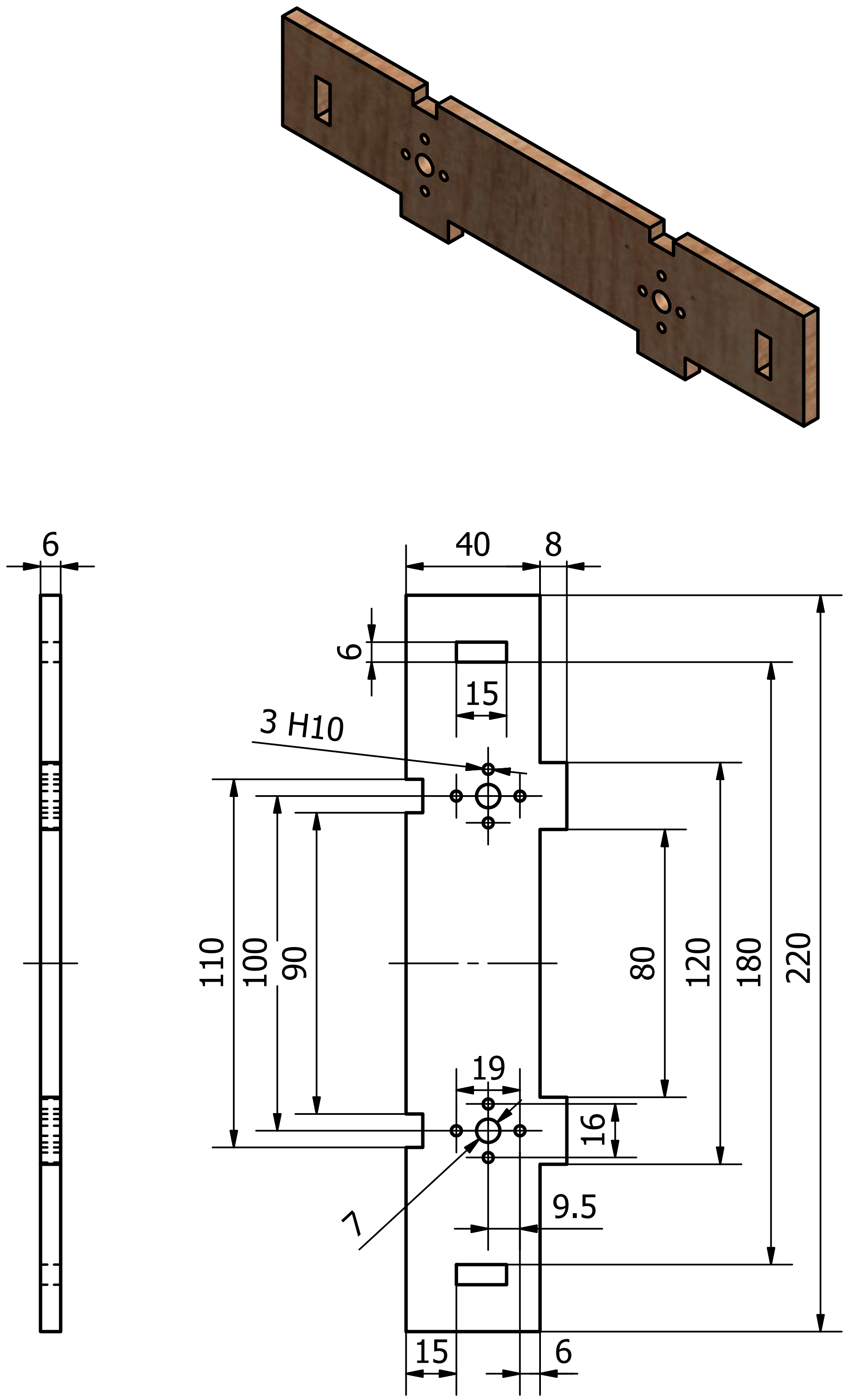


Designed by	Checked by	Approved by	Date		Date	Scale: 2:1
mhermoso					18/05/2016	

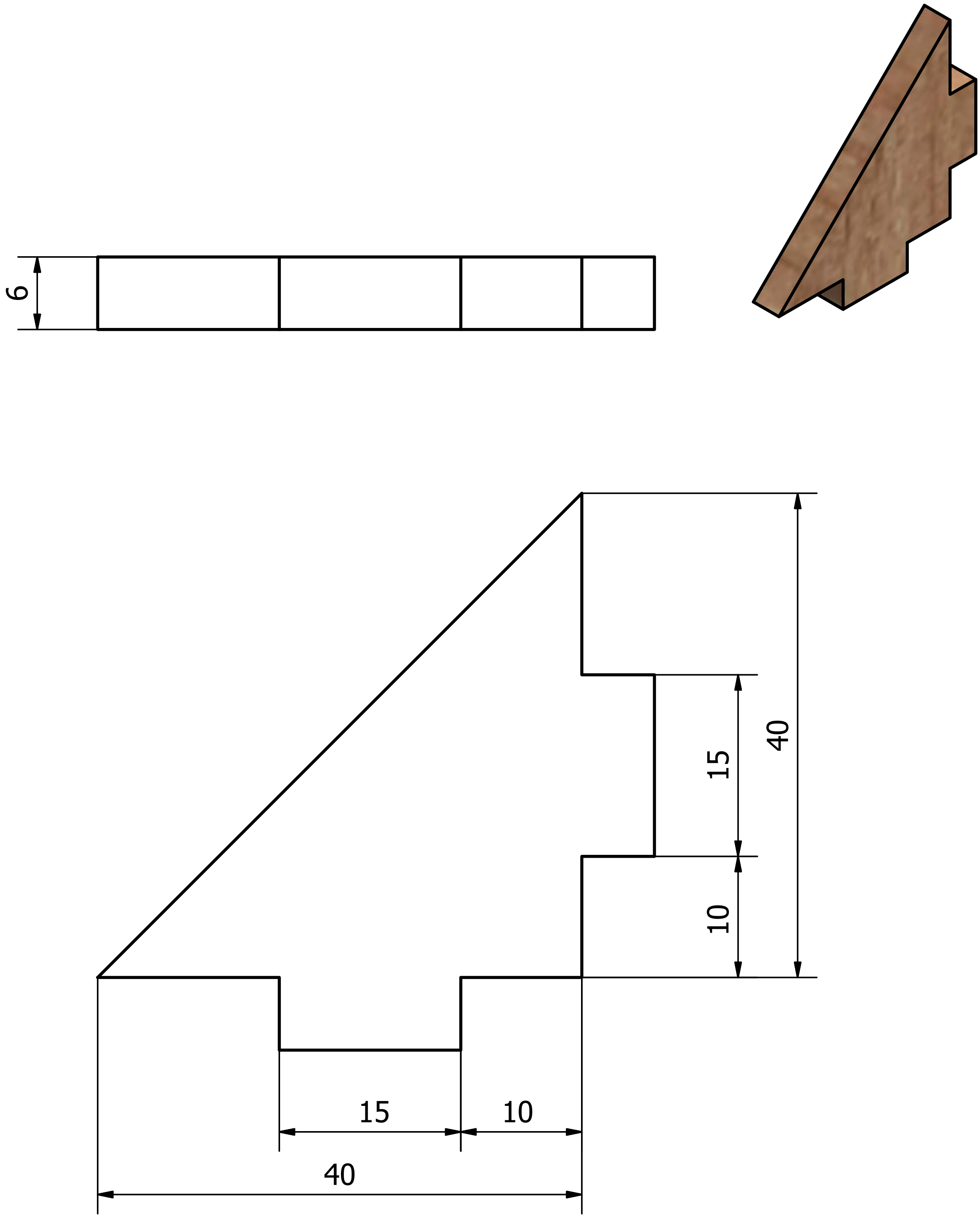
S4.9. Axis Pulley	Miguel Hermoso de Mendoza Sarasa		
	S4. Propellers Structure	Edition	Sheet 9 / 12



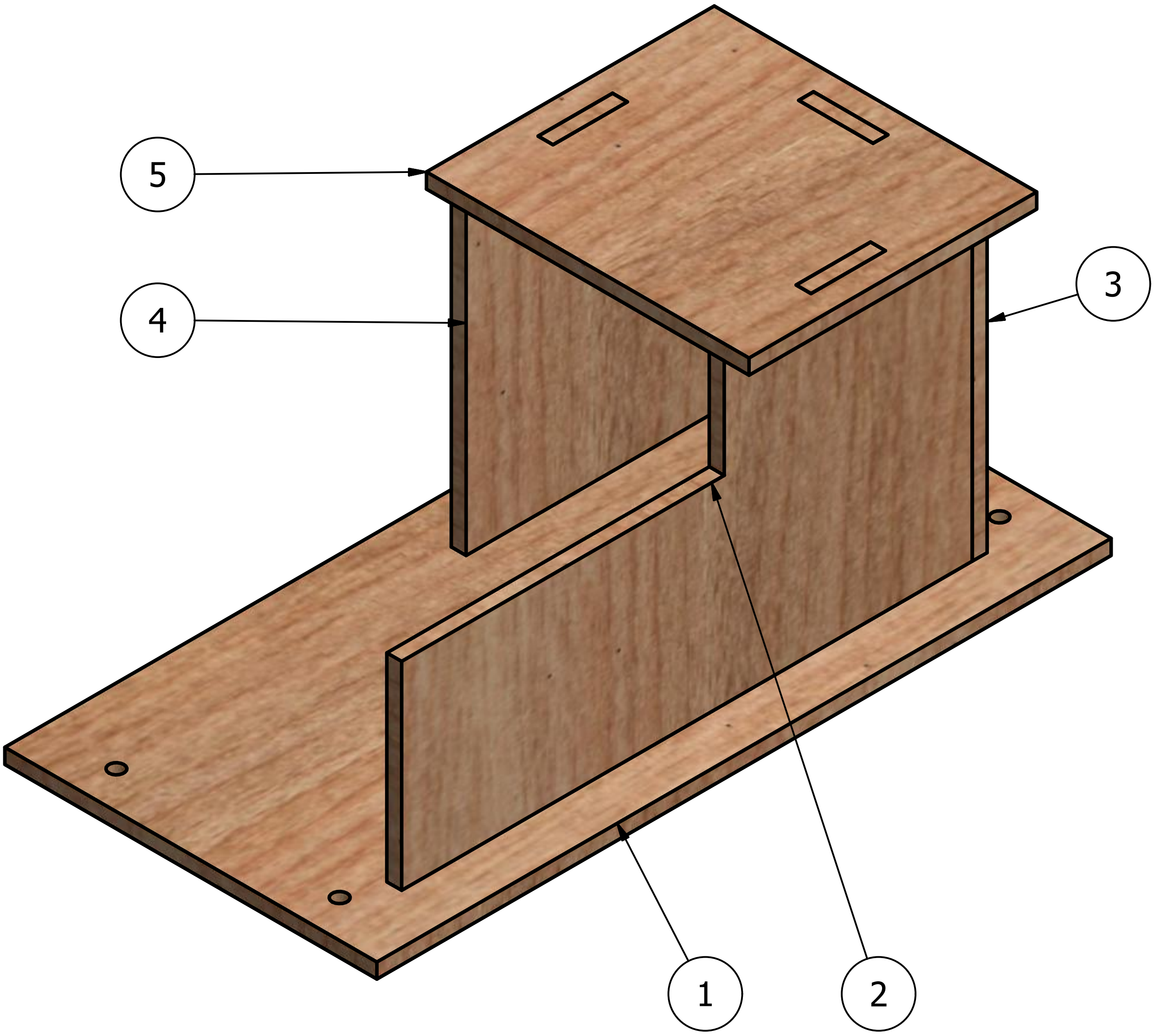
Designed by mhermoso	Checked by	Approved by	Date	Date 18/05/2016	Scale: 2:1
S4.10. Motor Pulley		Miguel Hermoso de Mendoza Sarasa			
		S4. Propellers Structure		Edition	Sheet 10 / 12



Designed by mhermoso	Checked by	Approved by	Date		Date 18/05/2016	Scale: 1:2
S4.11. Plate Supporting Motors			Miguel Hermoso de Mendoza Sarasa			
			S4. Propellers Structure		Edition	Sheet 11 / 12



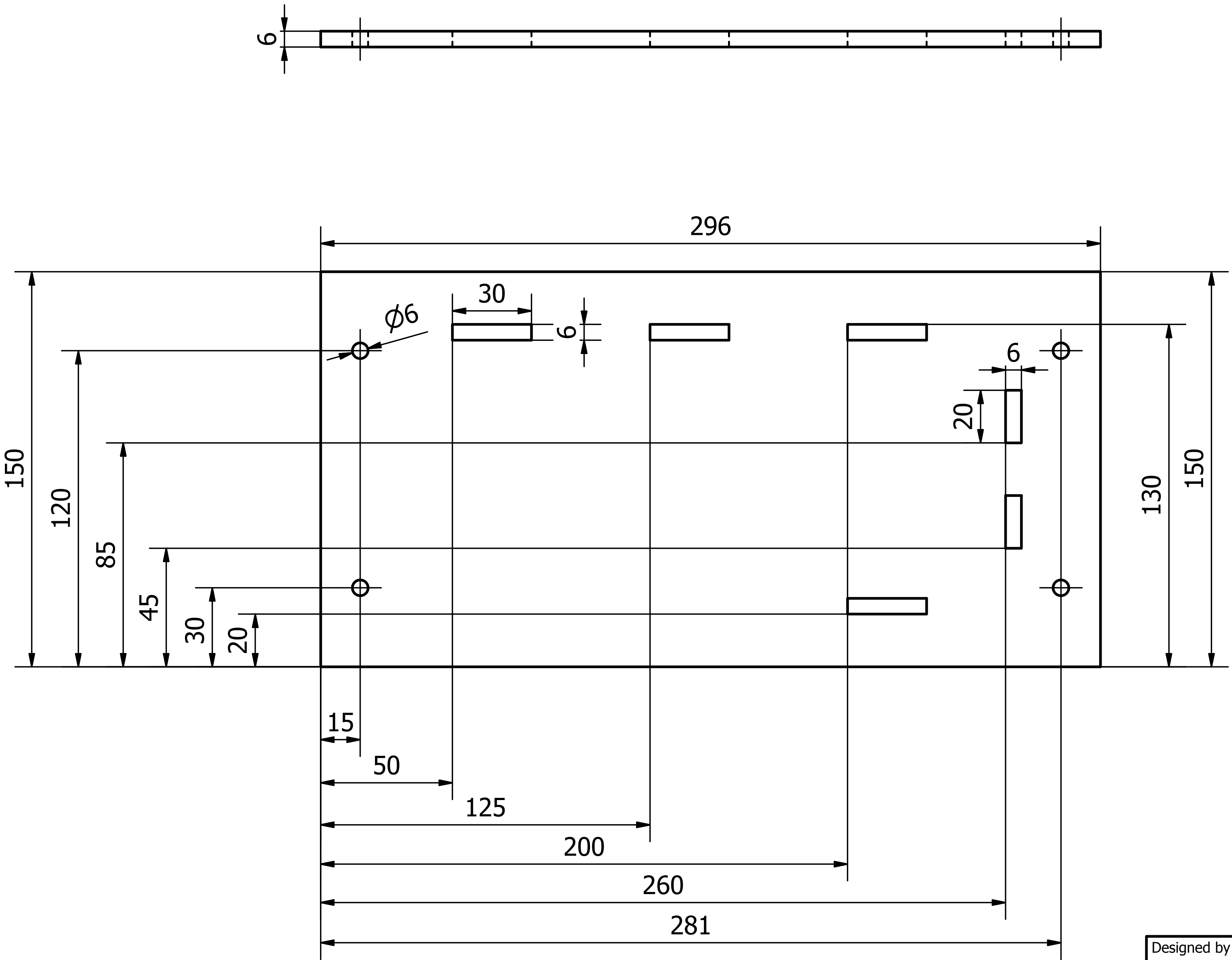
Designed by	Checked by	Approved by	Date	Date	Scale: 2:1
mhermoso				18/05/2016	
S4.12. Nerv	Miguel Hermoso de Mendoza Sarasa				
	S4. Propellers Structure	Edition		Sheet 12 / 12	



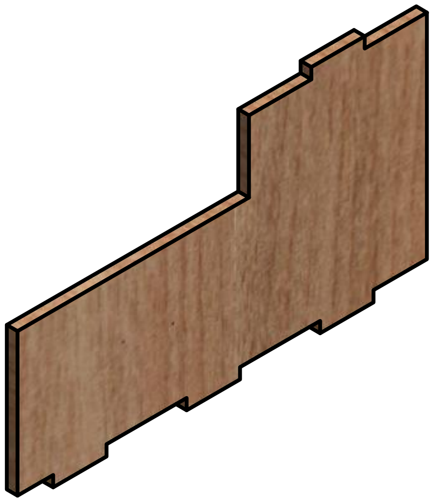
PARTS LIST			
ITEM	QTY	PART NUMBER	DESCRIPTION
1	1	Basis	
2	1	Wall 1	
3	1	Wall 2	
4	1	Wall 3	
5	1	Roof	

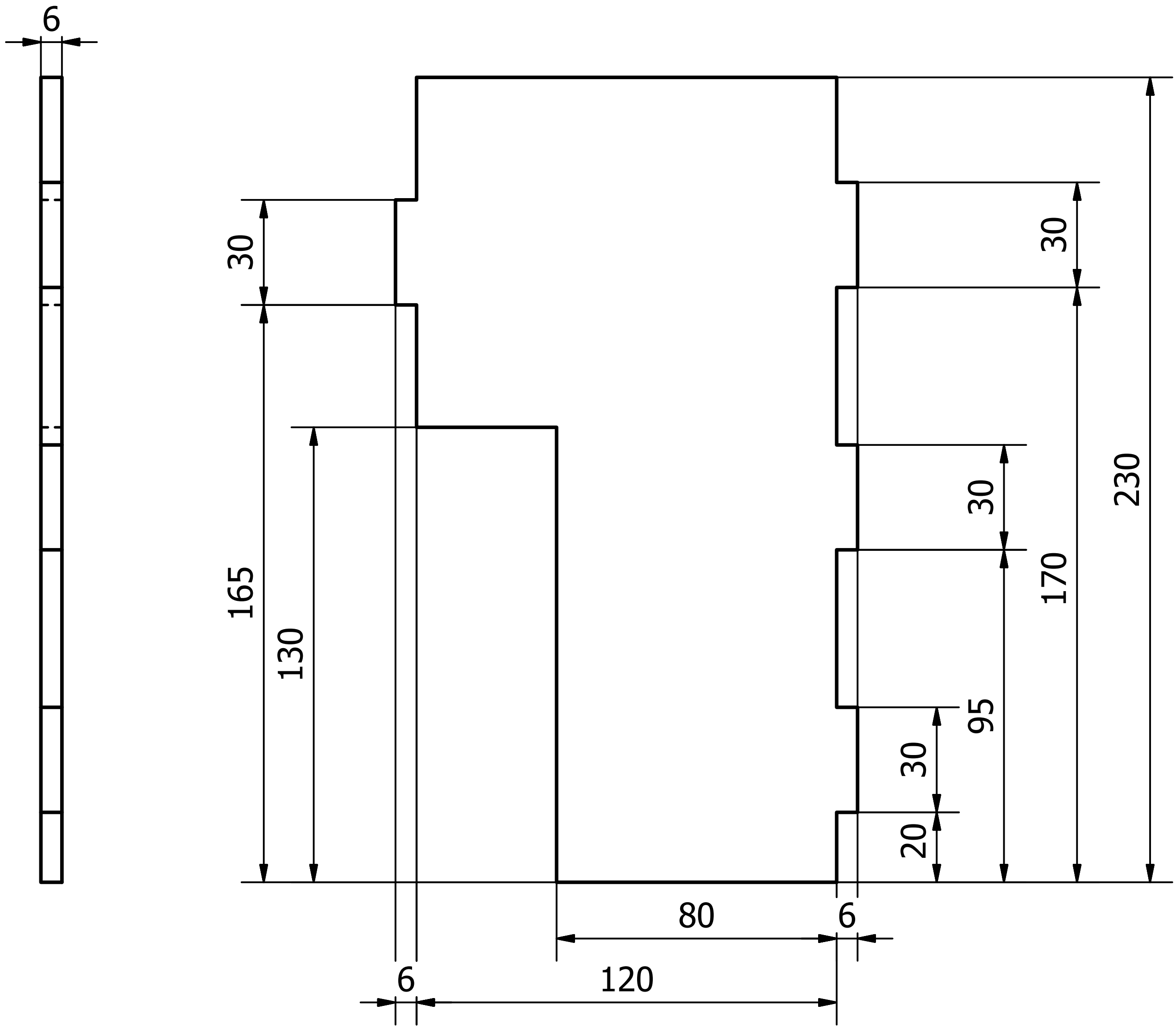
Designed by luchtvaart	Checked by	Approved by	Date	Date 6/06/2016	Scale: 1:2

V.I.V experiment set up	Miguel Hermoso de Mendoza Sarasa		
	S5. Electronics Support	Edition	Sheet 1 / 6



Designed by	Checked by	Approved by	Date	Date	Scale: 1:2
luchtvaart				6/06/2016	
S5.1. Basis	Miguel Hermoso de Mendoza Sarasa				
	S5. Electronics Support			Edition	Sheet 2 / 6





Designed by	Checked by	Approved by	Date		Date	Scale: 1:2
luchtvaart					6/06/2016	

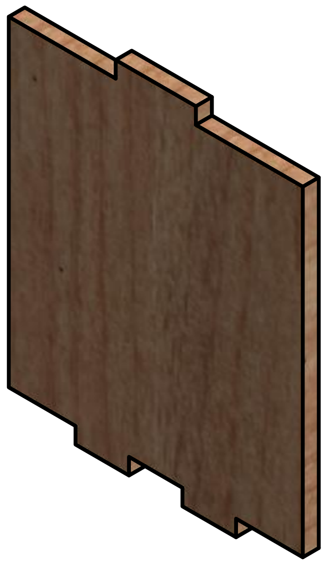
S5.2. Wall 1

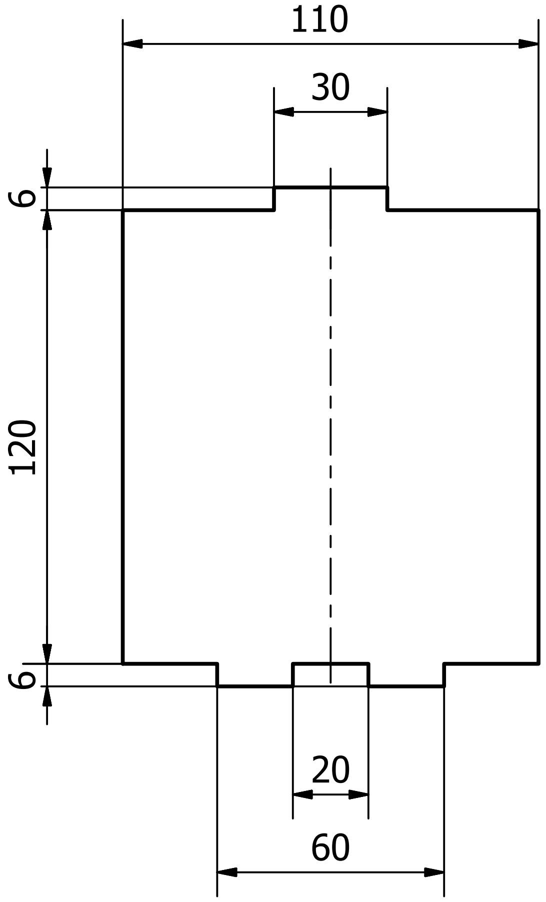
Miguel Hermoso de Mendoza Sarasa

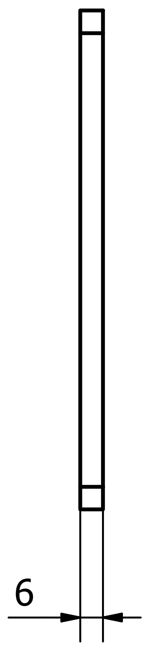
S5. Electronics Support

Edition

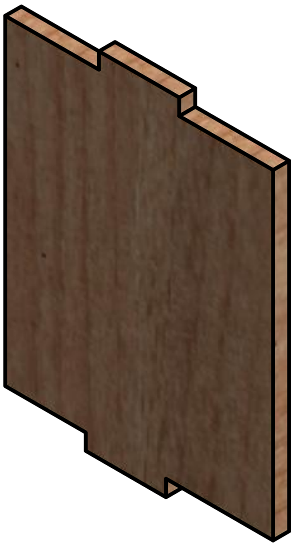
Sheet 3 / 6

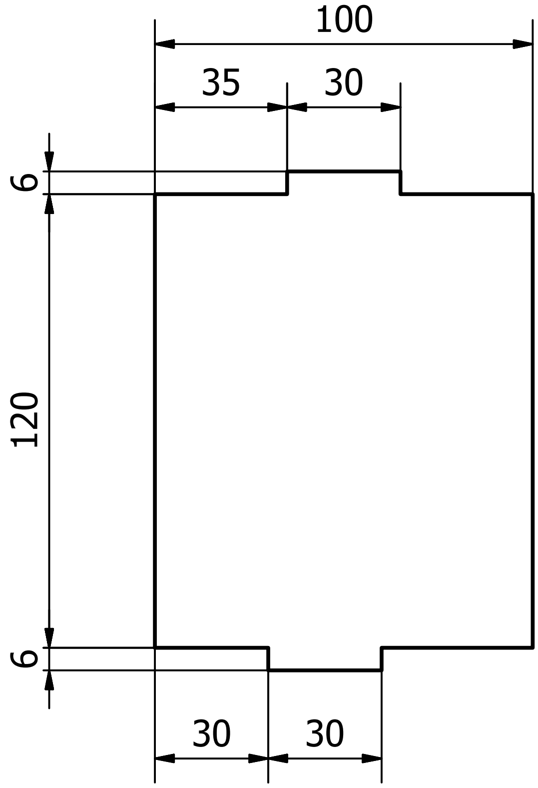


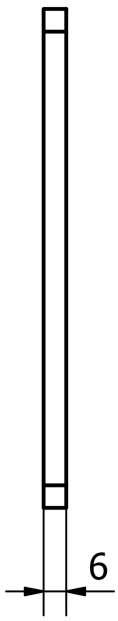




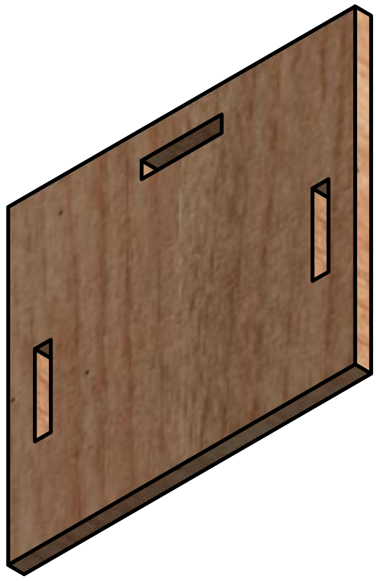
Designed by	Checked by	Approved by	Date		Date	Scale: 1:2
luchtvaart					6/06/2016	
S5.3. Wall 2		Miguel Hermoso de Mendoza Sarasa				
		S5. Electronics Support			Edition	Sheet 4 / 6

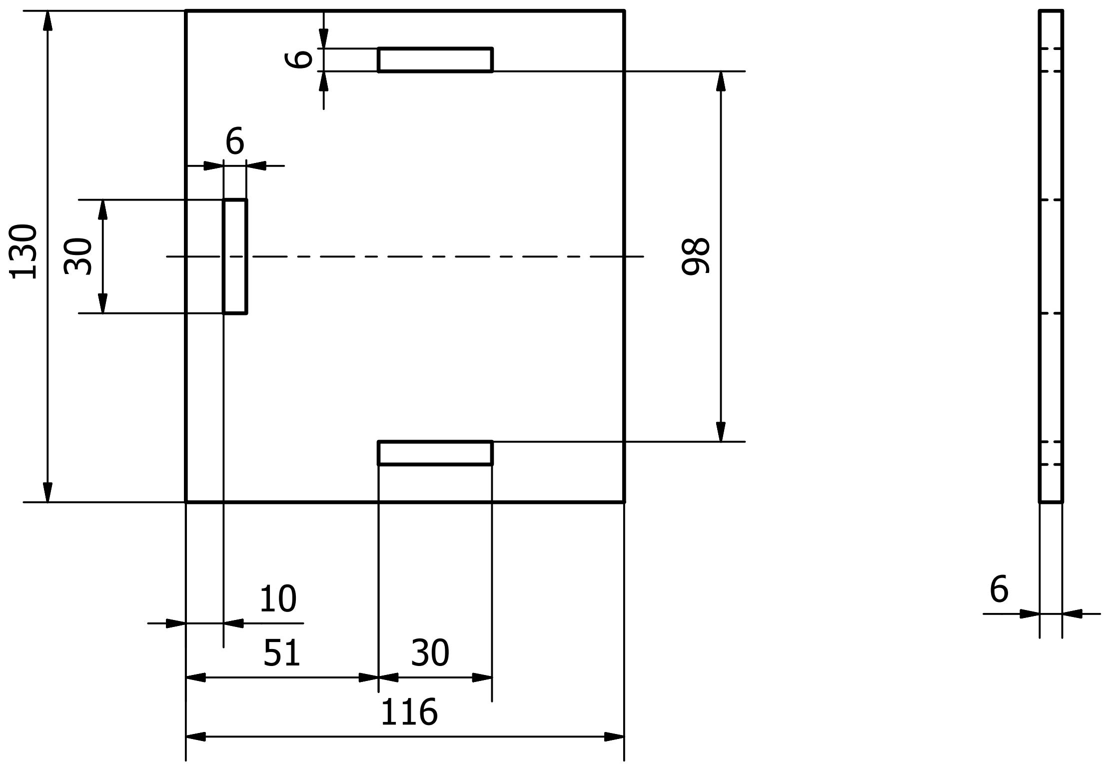






Designed by	Checked by	Approved by	Date		Date	Scale: 1:2
luchtvaart					6/06/2016	
S5.4. Wall 3		Miguel Hermoso de Mendoza Sarasa				
		S5. Electronics Support			Edition	Sheet 5 / 6





Designed by	Checked by	Approved by	Date		Date	Scale: 1:2
luchtvaart					6/06/2016	

S5.5. Roof

Miguel Hermoso de Mendoza Sarasa

S5. Electronics Support

Edition	Sheet
	6 / 6

Arduino commands

Brushless Motor automatic control

```
#include <Servo.h>

Servo myServo;


int potVal;

int ang;


int incomingByte = 0;   // for incoming serial data

int incomingByte2 = 0;

int incomingByte3 = 0;

int entero=0;

int decimal1=0;

int decimal2=0;

int PWM=0;

int Total=0;

float velocity=0;

int angle= 0;

int control=0;
```

```
void setup() {
  // put your setup code here, to run once:
        myServo.attach(9);
        Serial.begin(9600);
        myServo.write(angle);
        Serial.println("Please, select a frequency between 0 and 3 Hz");
        Serial.println("Write the entire part of your frequency: ");
        delay(2000);

}



void loop() {
  if (Serial.available() > 0) {
            // read the incoming byte:

            incomingByte = Serial.read();
            if (  incomingByte != 120  ) {
                     if (incomingByte >=48 && incomingByte <=51)
                     {
                     entero = incomingByte - 48;

                    // say what you got:
                   Serial.print("I 1 received: ");
                   Serial.println(entero, DEC);
                   delay(1000);
                   Serial.println("Write the decimal part of your frequency: ");
```

```
delay(5000);

    if (Serial.available() > 0 ) {
         // read the incoming byte:
         incomingByte2 = Serial.read();
         decimal1 = incomingByte2 - 48;

        // say what you got:
        Serial.print("I 2 received: ");
        Serial.println(decimal1, DEC);
        delay(1000);
          if (Serial.available() > 0) {
               // read the incoming byte:
               incomingByte3 = Serial.read();
               decimal2 = incomingByte3 - 48;

               // say what you got:
               Serial.print("I 3 received: ");
               Serial.println(decimal2, DEC);
               delay(1000);
          }
          else
          {
               decimal2=0;
               Serial.print("I 3 received: ");
               Serial.println(decimal2, DEC);
               delay(1000);
```

```
			}
		}
		else
		{
		 decimal1=0;
		 decimal2=0;
		 Serial.print("I 2 received: ");
		 Serial.println(decimal1, DEC);
		 Serial.print("I 3 received: ");
		 Serial.println(decimal2, DEC);
		 delay(1000);
		}

		if (entero == 3) {
		 Serial.println("Your frecuency can not be bigger than 3 Hz. Therefore:");
		 decimal1=0;
		 decimal2=0;
		}

	Total = entero * 100 + decimal1 * 10 + decimal2;
	float Total2 = entero * 100 + decimal1 * 10 + decimal2;
	float frequency = Total2/100;
	Serial.println("The frequecy selected is:  ");
	Serial.println(frequency, DEC);

	velocity= frequency * 0.03 / 0.2;
	Serial.println("The velocity of the water is:  ");
```

```
            Serial.println(velocity, DEC);

            delay(2000);


            Serial.println(" The motor is starting to spin. Write x if you want to stop the motor, or the new frequency of your cylinder if you want to change the water velocity ");

            ang = map(Total, 0, 300, 0, 29);

            angle = 150 + ang;

            Serial.print("Angle");

            Serial.println(angle);

            myServo.write(angle);

            delay(2000);

          }

          else

          {

           Serial.println("Please, select a frequency between 0 and 3 Hz");

           Serial.println("Write the entire part of your frequency: ");


          }}

 else

 {

  angle=0;

   myServo.write(angle);

  incomingByte=0;

  delay(1000);

  Serial.println("If you want the motor start rotating, write the entire part of your frequency: ");

 }

}

}
```

Stepper motor automatic control

```
int incomingByte = 0;   // for incoming serial data

int incomingByte2 = 0;

int incomingByte3 = 0;

int entero=0;

int decimal1=0;

int decimal2=0;

float tiempo=0;

int Total=0;

int control=0;

unsigned int n=0;

#define stepPin  4

#define Enable  2

#define Direction  3

unsigned int numberofsteps=9000000000;



void setup() {

      Serial.begin(9600);     // opens serial port, sets data rate to 9600 bps

      pinMode(stepPin, OUTPUT);

      pinMode(Enable, OUTPUT);

      pinMode(Enable, OUTPUT);

      Serial.println("Write the entire part of your frequency: ");

      delay(2000);

}

void loop() {
```

```
// send data only when you receive data:

if (Serial.available() > 0) {

    // read the incoming byte:
    incomingByte = Serial.read();
    if (incomingByte >=48 && incomingByte <=53)
    {
    digitalWrite(Enable, LOW);
    entero = incomingByte - 48;
    // say what you got:
    Serial.print("I 1 received: ");
    Serial.println(entero, DEC);
    delay(1000);
   Serial.println("Write the decimal part of your frequency: ");
   delay(5000);

        if (Serial.available() > 0) {
            // read the incoming byte:
            incomingByte2 = Serial.read();
              if (incomingByte2 >=48 && incomingByte2 <=57)
             {
               decimal1 = incomingByte2 - 48;

               // say what you got:
               Serial.print("I 2 received: ");
               Serial.println(decimal1, DEC);
```

```
            delay(1000);

            if (Serial.available() > 0) {

                // read the incoming byte:

                incomingByte3 = Serial.read();

                  if (incomingByte3 >=48 && incomingByte3 <=57)

              {

                decimal2 = incomingByte3 - 48;


                // say what you got:

                Serial.print("I 3 received: ");

                Serial.println(decimal2, DEC);

                delay(1000);

              }

           }

           else

           {

            decimal2=0;

            Serial.print("I 3 received: ");

            Serial.println(decimal2, DEC);

            delay(1000);

           }

    }

    }

    else

    {

     decimal1=0;

     decimal2=0;
```

```
            Serial.print("I 2 received: ");

            Serial.println(decimal1, DEC);+


            Serial.print("I 3 received: ");

            Serial.println(decimal2, DEC);

            delay(1000);

           }


        Total = entero * 100 + decimal1 * 10 + decimal2;

        float Total2 = entero * 100 + decimal1 * 10 + decimal2;

        float frequency = Total2/100;

        Serial.println("The frequecy selected is:  ");

        Serial.println(frequency, DEC);

        tiempo= ( 1/1 ) * 1000 / (frequency * 200);

        Serial.println("The step frequency is:  ");

        Serial.print(tiempo, DEC);

        Serial.println("  ms");

        delay(1000);

        Serial.println(" The motor is starting to spin. Write x if you want to stop or change its
velocity ");

        while( n < numberofsteps ) {


              digitalWrite(stepPin, HIGH);

              digitalWrite(stepPin, LOW);

              delay(tiempo);

              if (Serial.available() > 0)

              {

                   int control = Serial.read();
```

```
                if ( control == 120  )
                {
                numberofsteps=0;
                }
            n++;
            }
       }



       digitalWrite(Enable, HIGH);
       Serial.println(" The motor have stopped, Write the entire part of your frequency: ");
       numberofsteps=9000000000;
 }
 else
      {
       Serial.println("Please, select a frequency between 0 and 5 Hz");
       Serial.println("Write the entire part of your frequency: ");


      }
}
}
```